Sustainable Hydraulics in the Era of Global Change

Advances in Water Engineering and Research

Editors

Sébastien Erpicum
Benjamin Dewals
Pierre Archambeau
Michel Pirotton

CRC Press
Taylor & Francis Group
A BALKEMA BOOK

SUSTAINABLE HYDRAULICS IN THE ERA OF GLOBAL CHANGE

Taylor & Francis
Taylor & Francis Group
http://taylorandfrancis.com

PROCEEDINGS OF THE 4TH IAHR EUROPE CONGRESS, LIEGE, BELGIUM, 27–29 JULY 2016

Sustainable Hydraulics in the Era of Global Change

Advances in Water Engineering and Research

Editors

Sébastien Erpicum, Benjamin Dewals,
Pierre Archambeau & Michel Pirotton
Hydraulics in Environmental and Civil Engineering (HECE),
Department ArGEnCo, University of Liege (ULg), Liege, Belgium

CRC Press
Taylor & Francis Group
Boca Raton London New York Leiden

CRC Press is an imprint of the
Taylor & Francis Group, an **informa** business
A BALKEMA BOOK

Cover photos:
Front cover photo: Lanaye locks, situated across the Belgian-Dutch border in the north of Liege, are a key connexion in the European navigation network. The 4^{th} lock of Lanaye (225 m × 25 m, 14 m head difference), together with its pumping station and hydropower plant (2.3 MW), was awarded the 1^{st} IAHR Industry Innovation Award.

Courtesy: Parafly – http://www.parafly.be/
Service Public de Wallonie – http://voies-hydrauliques.wallonie.be/

CRC Press/Balkema is an imprint of the Taylor & Francis Group, an informa business

Typeset by MPS Limited, Chennai, India
Printed and bound in Great Britain by CPI Group (UK) Ltd, Croydon, CR0 4YY

Published by: CRC Press/Balkema
P.O. Box 11320, 2301 EH Leiden, The Netherlands
e-mail: Pub.NL@taylorandfrancis.com
www.crcpress.com – www.taylorandfrancis.com

ISBN: 978-1-138-02977-4 (Set of Hbk and CDROM)
ISBN: 978-1-4987-8149-7 (eBook PDF)

Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.)
© 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Table of contents

Hydrometeorological extremes, uncertainties and global change

Special session: Innovative solutions for adaptation of European hydropower systems in view of climate and market changes

Special session: Buoyancy-driven flows

Special session: Management of hydraulic systems by means of fuzzy logic

Taylor & Francis
Taylor & Francis Group
http://taylorandfrancis.com

Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.)
© 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Preface

This book is the Proceedings for the 4th European Congress of the International Association of Hydro-environment engineering and Research (IAHR), which gathered in Liege over 200 hydro-environment researchers and practitioners from all over Europe and beyond.

The overarching theme of the conference is entitled "Sustainable Hydraulics in the Era of Global Change", consistently with the pressing need to design and operate our water systems according to innovative standards in terms of climate adaptation, resource efficiency, sustainability and resilience. This grand challenge triggers unprecedented questions for hydro-environment research and engineering, particularly in relation with our increasingly urbanized societies. Addressing these issues requires a deep understanding of basic processes in fluid mechanics, heat transfer, surface and groundwater flow, among others. These are all themes widely covered by the book, which unveils latest research achievements and innovations relying on state-of-the-art modelling technologies and supported by the exponentially growing availability of data and computation power.

The 4th IAHR Europe Congress was organised by the research group *Hydraulics in Environmental and Civil Engineering* (HECE) of the University of Liege (ULg), supported by several key institutional and commercial sponsors, which are listed in the following pages. The Editors acknowledge the Authors and the members of the conference Committees, who were committed to ensuring a high scientific quality of the papers.

We are confident that the book will serve as a reference for professionals and decision-makers involved in various water-related sectors, such as hydraulic engineering, fluvial hydraulics, coastal engineering, water resources management and many more.

Benjamin DEWALS
Chair of the Organizing Committee

Sébastien ERPICUM
Chair of the International Scientific Committee

May 2016

Taylor & Francis
Taylor & Francis Group
http://taylorandfrancis.com

Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.)
© 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Committees

ORGANISING COMMITTEE

Chair

Benjamin DEWALS, *University of Liege (ULg), Liege, Belgium*

Members

Pierre ARCHAMBEAU, *University of Liege (ULg), Liege, Belgium*
Daniel BUNG, *Aachen University of Applied Sciences, Aachen, Germany*
Laurence DEFRÈRE, *University of Liege (ULg), Liege, Belgium*
Nadia ELGARA, *University of Liege (ULg), Liege, Belgium*
Sébastien ERPICUM, *University of Liege (ULg), Liege, Belgium*
Patrick HENDRICK, *Université Libre de Bruxelles, Brussels, Belgium*
Michel PIROTTON, *University of Liege (ULg), Liege, Belgium*
Holger SCHÜTTRUMPF, *RWTH Aachen University, Aachen, Germany*
Peter TROCH, *Ghent University, Ghent, Belgium*

IAHR EUROPE DIVISION LEADERSHIP TEAM

Chairs

Anton SCHLEISS (Chair), *Ecole Polytechnique Fédérale de Lausanne (EPFL), Lausanne, Switzerland*
Corrado GISONNI (Vice Chair), *2nd University of Naples, Naples, Italy*
Aronne ARMANINI (Past Chair), *Universita degli Studi di Trento, Trento, Italy*

Members

Silke WIEPRECHT, *University of Stuttgart, Germany*
Alessandra CROSATO, *UNESCO-IHE Institute for Water Education, the Netherlands*
Francisco TAVEIRA PINTO, *Universidade Do Porto, Portugal*
Manuel GÓMEZ VALENTÍN, *Universidad Politécnica de Cataluña – EHMA, Spain*
Bettina BOCKELMANN-EVANS, *Cardiff School of Engineering, United Kingdom*
Jean-Paul CHABARD, *Electricité de France (EDF), France*
Benjamin DEWALS, *University of Liege (ULg), Belgium*
Markus AUFLEGER, *Innsbruck University, Austria*
Peter RUTSCHMANN, *Technische Universität München, Germany*
Pawel M. ROWINSKI, *Polish Academy of Sciences, Poland*
Ian GUYMER, *University of Warwick, United Kingdom*
Mark RIEDER, *T.G. Masaryk Water Research Institute, Czech Republic*

INTERNATIONAL SCIENTIFIC COMMITTEE

Chair

Sébastien ERPICUM, *University of Liege (ULg), Liege, Belgium*

Members

Jochen ABERLE, *Norway*
Claudia ADDUCE, *Italy*
Pierre ARCHAMBEAU, *Belgium*
Markus AUFLEGER, *Germany*
Francesco BALLIO, *Italy*
Koen BLANCKAERT, *Switzerland*
Bettina BOCKELMANN-EVANS, *UK*
Didier BOUSMAR, *Belgium*
Daniel BUNG, *Germany*
Benoît CAMENEN, *France*
Jean-Paul CHABARD, *France*
Hubert CHANSON, *Australia*
Alessandra CROSATO, *The Netherlands*
Christophe CUDENNEC, *France*
Alain DASSARGUES, *Belgium*
Rita de CARVALHO, *Portugal*
Tom DE MULDER, *Belgium*
Benjamin DEWALS, *Belgium*
Andreas DITTRICH, *Germany*
Dejana DJORDJEVIC, *Serbia*
Rui FERREIRA, *Portugal*
Corrado GISONNI, *Italy*
Manuel GOMEZ VALENTIN, *Spain*
Philippe GOURBESVILLE, *France*
Massimo GRECO, *Italy*
Ian GUYMER, *United Kingdom*
Patrick HENDRICK, *Belgium*
Vlassios HRISSANTHOU, *Greece*
Juha JARVELA, *Finland*
Katinka KOLL, *Germany*
Elpida KOLOKYTHA, *Greece*
Jorge MATOS, *Portugal*
Pedro MANSO, *Switzerland*
Jaak MONBALIU, *Belgium*
Michele MOSSA, *Italy*
Vladimir NIKORA, *United Kingdom*
Mario OERTEL, *Germany*
Stefano PAGLIARA, *Italy*
André PAQUIER, *France*
Gareth PENDER, *United Kingdom*
Fernando PEREIRA, *Belgium*
Michael PFISTER, *Switzerland*
Francisco PINTO, *Portugal*
Michel PIROTTON, *Belgium*
Sebastien PROUST, *France*
Jaromir RIHA, *Czech Republic*
Nicolas RIVIERE, *France*
Pawel ROWINSKI, *Poland*
Nils RUTHER, *Norway*
Paul SAMUELS, *United Kingdom*
Hubert SAVENIJE, *The Netherlands*
Anton SCHLEISS, *Switzerland*
Holger SCHUTTRUMPF, *Germany*
Koji SHIONO, *United Kingdom*
Sandra SOAREZ-FRAZAO, *Belgium*
Thorsten STOESSER, *United Kingdom*
Mutlu SUMER, *Denmark*
James SUTHERLAND, *United Kingdom*
Simon TAIT, *United Kingdom*
Erik TOORMAN, *Belgium*
Peter TROCH, *Belgium*
Blake TULLIS, *United States*
Wim UIJTTEWAAL, *The Netherlands*
Pierre-Louis VIOLLET, *France*
Patrick WILLEMS, *Belgium*
Weiming WU, *United States*
Gerald ZENZ, *Austria*

Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.)
© 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Support from institutions and scientific societies

The Congress is sponsored by the *International Association for Hydro-Environment Engineering and Research* (IAHR) and co-sponsored by:

- the *International Association of Hydrological Sciences* (IAHS),
- and *Société Hydrotechnique de France* (SHF).

The Congress is also supported by

- the *University of Liege* (ULg),
- the Belgian research agency *Fund for Scientific Research* (FRS-FNRS).

Taylor & Francis
Taylor & Francis Group
http://taylorandfrancis.com

Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.)
© 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Sponsors

MAIN SPONSORS

CO-SPONSORS

Keynotes

Taylor & Francis
Taylor & Francis Group
http://taylorandfrancis.com

Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.)

Co-development of coastal flood models: Making the leap from expert analysis to decision support

B.F. Sanders, A. Luke, J.E. Schubert, H.R. Moftakhari & A. AghaKouchak
Department of Civil and Environmental Engineering, University of California, Irvine, USA

R.A. Matthew, K. Goodrich, W. Cheung, D.L. Feldman, V. Basolo & D. Houston
Department of Planning, Policy and Design, University of California, Irvine, USA

K. Serrano
Sustainability Initiative, University of California, Irvine, USA

D. Boudreau & A. Eguiarte
Tijuana River National Estuarine Research Reserve, USA

ABSTRACT

Metric resolution "urban" flood models are emerging as powerful tools for analyzing and communicating flood risk at fine spatial and temporal scales which align with personal awareness of geographical areas, and differentiate across individual assets vulnerable to flooding such as homes, businesses, industrial facilities, health care facilities, schools, parks, places of worship, and environmental resources. A combination of trends such as urbanization, intensification of the hydrologic cycle, and higher sea levels portends a significant increase in urban flooding hazards (Hanson et al. 2011). Furthermore, development pressures in many communities mean greater willingness to build in high hazard areas such as floodplains. Urban flood models have potential to provide valuable information about the impacts of development decisions and flood mitigation measures on flood risk, and to support planning for and responding to severe flooding events. However, there is a dearth of knowledge regarding how to transform dense spatiotemporal flood model output data into information that can be used by decision makers. To develop new and improved flood modeling systems, engineers also need to deepen understanding of flooding as a coupled human-water system (Sivapalan et al. 2012).

The Flood Resilient Infrastructure and Sustainable Environments (FloodRISE) project funded by the US National Science Foundation (#1331611) has resulted in the co-development of metric resolution flood models (e.g., Gallien et al. 2011, 2014) in three communities: Newport Beach, Calif.; Tijuana River Valley, Calif.; and Los Laureles in Tijuana, Baja Calif. Co-development of flood models refers to a two-way communication and development process involving the research team and personnel working and living in the study areas, including residents, government officials, emergency managers, civil society groups and business leaders. Activities include in-depth interviews to gather qualitative data about flooding, a formal field survey to gather information about community awareness of and preparedness for flooding, and focus groups to deepen understanding of the decision-points that can be served by flood models and the map formats that best serve decision-making needs. Examples include maps of the 100-year flood depth, the annual probability of flooding, flood intensity (i.e., depth times velocity), and future vs. present flood risk.

Focus groups revealed strong preferences for visualized flood risk information based on decision-making needs. Officials responsible for city planning and regulatory compliance were most interested in the 100-year flood zone presumably as a consequence of flood insurance policy in the USA. Water rescue personnel were most interested flooding intensity and valued greater awareness of the geographical scope of swift water hazards. Civil society groups were most interested the frequency and duration of flooding to assess potential flood damage to local ecosystems. Collectively, results suggest that a diverse set of decision-support needs can be met by a single, fine-scale modeling tool when model output is transformed into a set of maps that communicate different aspects of the flood, and address decision points relevant to diverse users. Further, we conjecture that the process of co-developing the flood model with stakeholders builds confidence in the model among both experts and community members and also creates a solid foundation to plan and evaluate flood risk interventions.

REFERENCES

Gallien, T.W., Sanders, B.F. and Flick R.E. 2014. Urban coastal flood prediction: Integrating wave overtopping, flood defenses, and drainage. Coastal Engineering, 91, 18–28.

Gallien, T.W., Schubert, J.E. and Sanders B.F. 2011. Predicting tidal flooding in urbanized embayments: A modeling framework and data requirements. Coastal Engineering, 58(6), 567–577.

Hanson, S., Nicholls, R., Ranger, N., Hallegatte, S., Corfee-Morlot, J., Herweijer, C., Chateau, J. 2011. A global ranking of port cities with high exposure to climate extremes Climatic Change, 104(1): 89–111.

Sivapalan, M., Savenije, H. H. G., & Blöschl, G. 2012. Socio-hydrology: A new science of people and water. *Hydrological Processes*, 26(8), 1270–1276.

Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.)
© 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Challenges and opportunities for research and technological innovation in the water sector throughout Europe

P. Balabanis
European Commission, Directorate-General for Research & Innovation, Belgium

ABSTRACT

In today's world where intensive use of the world's resources puts pressure on our planet and threatens economic prosperity, growth and jobs, innovative solutions are needed to help us using our resources more efficiently and anticipate more complex demands. Global change, population increase, urbanization are particularly challenging the sustainability of water resources. For the last three consecutive years, the Global Risk report of the World Economic Forum puts water systematically as one of the highest risks that could undermine economic growth.

Citizens, societies, agriculture and industries will increasingly need innovative solutions to meet the need of using water in a more efficient and effective way. Innovative thinking and smarter use of innovation have the potential to bring new solutions quickly and efficiently to the market while responding to the needs of end users in urban, rural and industrial areas. Innovative solutions to water related challenges can directly support wider environmental objectives such as protecting our natural capital and ecosystems, and the biodiversity that supports these. In addition, solutions with regard to drinking water and waste water treatment are to the benefit of public health, which in turn will generate significant savings. Furthermore, solutions to improve protection of, and in, flood-prone areas will enhance public safety and prevent potential economic losses.

Water has been an important activity in successive European Research and Development Framework Programmes over the last decades. Horizon 2020, the current 2014–2020 European Union funding programme for research and innovation, expands the scope of previous water research activities, by addressing the whole chain of research and innovations with the aim of unlocking the innovation potential in the field of water management.

In line with Horizon 2020 objectives, a dedicated focus area "Water innovation: boosting its value for Europe" has been identified in the first Horizon 2020 work programme (2014–2015). This focus area addressed demonstration and market replication activities for eco-innovative, integrated and cross-sectoral solutions for water management. In the second Horizon 2020 work programme actions to boost water innovation for Europe and beyond will be addressed in the areas of the circular economy, sustainable cities, climate services, territorial resilience etc., as well as in other parts of Horizon 2020. Actions strengthening the role of water in the circular economy will be particularly promoted. In this context large scale demonstration/pilot projects, exhibiting a sufficient level of novelty and progress with respect to the state of the art and aiming to implement and test new technological and non-technological solutions through first-of-a-kind experimental development under real solutions, are foreseen. These project should also act as a way to attract most interest from innovators and innovation users (e.g. industries, financial actors, academia, research, private or public entities, regions, cities, citizens and their organizations, etc.), thus helping to unlock additional public/private investments in the water sector and strengthen complementary or synergies with other relevant EU funding mechanisms and initiates, especially, the European Innovation Partnership and Joint Programming Initiative on Water.

REFERENCES

European Commsion, 2011. Recommendation on the research joint programming initiative Water challenges for a changing world, Brussels, OJ C 317, p. 1–3.

European Commission, 2012. European Innovation Partnership water, Strategic Implementation Plan, Brussels.

European Commission, 2013. Regulation of the European Parliament and European Council of 11 December 2013 establishing Horizon 2020 – the Framework Programme for Research and Innovation (2014-2020), Brussels, OJ L 347, p. 81–1030.

World Economic Forum, 2016. The Global Risks Report 2016. 11th Edition, Geneva.

References

Hydro-environment and eco-hydraulics

wedge deformation near the source region of the

2011 tohoku-oki earthquake. Geophysical Research Let

ters 38(7), n/a–n/a. L00G05.

Kanamori, H. (1972). Mechanism of tsunami earthquakes.

Phys. Earth Planet. Interiors 6, 346–359.

Koketsu, K., Y. Yokota, N. Nishimura, Y. Yagi, S. Miyazaki,

K. Satake, Y. Fujii, H. Miyake, S. Sakai, Y. Yamanaka,

& T. Okada (2011). A unified source model for the

2011 tohoku earthquake. Earth and Planetary Science

Letters 310(34), 480 – 487.

Nosov, M. (2014). Tsunami waves of seismic origin: The

modern state of knowledge. Izvestiya, Atmosperic and

Oceanic Physics 50(5), 474–484.

Satake, K., Y. Fujii, T. Harada, & Y. Namegaya (2013, May).

Time and space distribution of coseismic slip of the 2011

tohoku earthquake as inferred from tsunami waveform

data kenji satake. Bulletin of the Seismological Society of

America 103(2B), 1473–1492. Shao, G., X. Li, C. Ji, & T. Maeda (2011). Focal mechanism and slip history of the 2011 mw 9.1 off the pacific coast of tohoku earthquake, constrained with teleseismic body and surface waves. Earth, Planets and Space 63(7), 559–564. Todorovska, M. I. & M. D. Trifunac (2001). Generation of tsunamis by a slowly spreading uplift of the sea floor. Soil Dynamics and Earthquake Engineering 21, 151–167. Yates, M. L. & M. Benoit (2015). Accuracy and efficiency of two numerical methods of solving the potential flow problem for highly nonlinear and dispersive water waves. International Journal for Numerical Methods in Fluids 77(10), 616–640. Zakharov,

V. E. (1968, March). Stability of periodic waves of finite amplitude on the surface of a deep fluid. Journal of Applied Mechanics and Technical Physics 9, 190–194.
Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Free surface flow through homogeneous bottom roughness

H. Romdhane & A. Soualmia

National Agronomic Institute of Tunisia, University of Carthage, Tunisia

L. Cassan & D. Dartus

Institute of Fluid Mechanics of Toulouse, France

ABSTRACT: The flow in channels and rivers is strongly influenced by the roughness of the channel bottom,

by the presence of vegetation or other obstacles. In this context, an experimental study was conducted in the

laboratory of the Institute of Fluid Mechanics of Toulouse "IMFT". The experimental design is a rectangular

channel 4 m long and 0.4 m wide and 0.8 m deep. The bottom has a homogeneous roughness contrast (installation

of a mat). For the measurement of speed, the channel is equipped with a fast camera. The originality of our work

lies in the application of a PTV particle tracking technique (Particle Tracking Velocimetry). It is a non-intrusive

measurement technique to measure instantaneous speed two-dimensional fields in stationary and unsteady flows.

It involves seeding the flow through reflective particles, with the same density as the fluid, which will be lit using

a lighting plan. Analysis of the results is by processing images taken by the camera using Quick Matlab that is

suitable for our case study. The obtained results show a depression of the maximum speed below the free surface.

This behavior indicates a delay of the flow in the vicinity

of the free surface and this is a direct consequence

of the presence of secondary flows in these areas. These
conducted measurements allowed the determination of

wall parameters which are fundamental.

1 INTRODUCTION

The flow in channels and rivers is strongly influenced

by the roughness of the channel bottom, by the pres

ence of vegetation or other obstacles (Franc Vigié,

2005).

Therefore, an experimental study was conducted

at the laboratory of the Institute of Fluid Mechanics

of Toulouse (IMFT). The experiments were carried

out in a rectangular inclinable flume with transverse

homogeneous roughness in the channel bottom.

A one-dimensional Particle Tracking Velocimetry

(PTV) technique has been developed to provide the

velocity fields. One major advantage of this technique

is that the flow field structures can be examined at a

prescribed instant of time in total. However, this mea

surement method has also some limitations such as;

this technique lies exactly in its ability to gather large

amounts of flow field information in the form of image

data and usually requires the analysis of a large number

of images, (Adamczyk & Rimai, 1988).

By means of a fast camera, we determined the

transverse evolution of the fields of average velocity.

Several studies have been conducted in order to

investigate the flow over rough bottom. We can quote

some of these works: Chouaib Labiod (2005) realized

experiments with a mode of setting up of the bars on

the bottom of channel.The strips have a length equal to

the channel width and thus the roughness of the bottom is homogeneous; Emma Florens (2010) also realized experiments with macro roughness on the channel bottom; Zaouali Sahbi (2008) studied the structure and the modeling of free surface flows in channels with inhomogeneous roughness. The roughness channel has been the goal of a number of literatures, such as: A. A. Adamczyk and Rimai (1988); Baki, et al (2014); Brevis, et al (2014); Cassan, (2014). 2 EXPERIMENTAL DESIGN 2.1 Description of the open channel The experimental device (Fig 1) is composed of a rectangular channel made of glass, open pit having length of 4 m, a height equal to 0.8 m and a width of 0.4 m. The slope of the channel being adjustable, and varies between 0 and 6%. Circulation loop (stable) water is provided by an electric pump providing a maximum throughput of 20 l/s. This pump delivers water through a pipe from a downstream tank from the channel to another upstream. The control of the water level is done using a valve downstream of the channel and the flow control is done with a guillotine valve. Flow rates were measured using electromagnetic flow meters KRHONE with an accuracy of 0.5%.

Figure 1. The Channel and the annexes.

2.2 Roughness configuration

On the merits, initially smooth, we stuck in the lon

gitudinal direction of the flow equidistant roughness

lurking, with 8 mm high, distributed throughout the

channel.

Thus, we achieve a high uniform roughness back

ground, object of our study.

2.3 Measurement Technology: PTV (Particle Tracking

Velocimetry)
In this study, the measuring means developed and used corresponding to a particle tracking technique (Parti cle Tracking Velocimetry) using a fast camera. This is a non-intrusive measurement technique for measuring two-dimensional fields of instantaneous velocities in unsteady and steady flows. It consists in inoculating the flow using reflective particles, having a diameter about 2 mm and the same density as the fluid, which will be illuminated with an illumination plane (Cassan et al, 2013).
A fast camera (1024 * 1280 pixels) allows you to view the free surface of a pattern by ombroscopy, placed face a LED lighting system to differentiate the air from the water. The acquired image sequence sets the gray scale particles which, after treatment, will have access to the two components of the velocity vector in the plan.
Compared to other measuring methods, the PTV is an effective way to access instantly to different spatial scales of a turbulent flow in the same plane. Indeed, while the LDV, for example, requires a lot of careful movement of the equipment to obtain a high spatial resolution in a measuring plane in question, PTV pro vides, in the same plane, the information at the same

instant in each point of space, (Adamczyk & Rimai,

1988).

A series of 5000 images is taken for each flow rate.

The averaged time value is determined by calculat

ing the averaged signal for each pixel. This enabled us

to calculate a mean for the water depth in the trans

verse direction. The free surface is identified by the

minimum signal. A mean water depth of the pattern is

then derived by integrating the free surface in the lon

gitudinal direction. Flow rates were measured using Figure 2. The channel bottom roughness. KRHONE electromagnetic flow meters with an accuracy of 0.5%. The slopes of the channel tested are: 1% and 2%. The flow rates for each slope are respectively: 5, 10, 15 l/s. By determining the displacement of a particle between two consecutive images, we can measure the velocity. Analysis is performed on a large number of particle detection. By grouping the measures in areas 20, 40 or 50 pixels, we obtain a cartography of the averaged transversely velocity field. The Stokes number is a dimensionless number used in fluid dynamics to study the behavior of a particle in a fluid. It represents the ratio between the kinetic energy of the particle to the energy dissipated by friction with the fluid. Is defined as follows: Where: ρ p : Density of the particle (1 kg/m 3) d p : Characteristic length of the particle (0.0008 m) v: Cinematic Fluid velocity μ: Dynamic viscosity of the fluid (10 kg/m.s) L c : Characteristic length (m) This number is used to determine the behavior of a particle in a fluid encountered an obstacle and in particular whether the particle will circumvent the obstacle (if <1) by following the movement of fluid or if it will percolate the obstacle (if >1). In our case it is: The Stokes number is small, so it is in a case where the particles follow the water flow.

Figure 3. The PTV equipments in the IMFT laboratory.

2.4 Definition of parameters to be measured

In these experiments we measured the following parameters:

h (m): Water depth in the channel

Q (l/s): Flow rate transited

U (m/s): Velocity component in the X direction

V (m/s): Velocity component in the Z direction

K (m^2/s^2): Kinetic energy

$u'w'$ (m^2/s^2): Turbulent shear stress

Where:

X: horizontal axis

Z: vertical axis

3 EXPERIMENTAL RESULTS

3.1 Description of the experimental tests

Several tests were made with the fast camera; we

worked with different frequency generator, different

number of images per sequence (10000 and 5000

images). To claim that this is the maximum frequency

given by the generator this gives the most accurate

results. Secondly, we choose to work with 5000 images Table 1. The Stokes number for the different runs. Slope 1% Slope 2% Q1 (5 l/s) 2 10^{-7} 3 10^{-7} Q2 (10 l/s) 3 10^{-7} 5 10^{-7} Q3 (15 l/s) 4 10^{-7} 6 10^{-7} Table 2. Definition of the experimental runs. slope 1% Slope 2% Flow Water Flow Water rates depth rates depth (l/s) (cm) (m^3/s) (cm) Run 1 Q1 = 5 h1 = 4.5 Q1 = 5 h1 = 3 Run 2 Q2 = 10 h2 = 6 Q2 = 10 h2 = 4 Run 3 Q3 = 15 h3 = 7 Q3 = 15 h3 = 5 per sequence because with 10000 images the recording and processing are slower. In the following tests were therefore kept a number of image 5000 im/Seq and we worked with the maximum frequency of 360 Hz. 3.2 Dimensionless results The figures below shows the dimensionless results were $u*$ it is the $u*$ determined by a theoretical method: On figure (Fig.4), was drawn the vertical profiles of the mean longitudinal velocity, above the rough area, for different flow rates.

It is found that more the flow rate became higher; more increases the velocity U, which is quite expected. Also, you will notice the same thing for the transverse velocity V, the turbulent kinetic energy K and the turbulent shear stress u ′ w ′ , an increasing with the flow rate increasing. We also note that, on these profiles, a depression of the maximum velocity below the free surface. In fact this behavior indicates a retardation of the flow in the vicinity of the free surface, which is a direct consequence of the presence of secondary flows in these areas (Labiod, 2005). These vertical profiles confirm the in-situ and literature observations (Baki & al. 2013).These measurements with PTV, show that transverse velocity V (Fig.5) near the surface is lower and maximum near the bottom. These measures must be taken with caution as the particle detection in the area near the free surface is difficult (Cassan & al, 2013). Fig.6 shows that the profiles of the turbulent shear stress u ′ w ′ often exhibit deviations from the expected

Figure 4. Vertical profile of the longitudinal velocity U for

the different flow rates (Q1 = 5 l/s; Q2 = 10 l/s; Q3 = 15 l/s)

and the different slopes (a): slope 1% (b): slope 2%

Figure 5. Vertical profile of the transverse velocity V for

the different flow rates (Q1 = 5 l/s; Q2 = 10 l/s; Q3 = 15 l/s)

and the different slopes (a): slope 1% (b): slope 2%. Figure 6. Vertical profile of the turbulent shear stress u ′ w ′ for the different flow rates (Q1 = 5 l/s; Q2 = 10 l/s; Q3 = 15 l/s) and the different slopes (a): slope 1% (b): slope 2%. linear profile of parallel flow: it is the most significant sign of the presence of secondary flows and their impact on transportation of longitudinal movement quantity (Zaouali, 2008). In the wall region, the shear stress is greater at the level of the rough area. This is a direct effect of the roughness. Away from the wall, the situation is reversed following the adjective transport turbulence, by descendant’s flows, from low production zones (free surface) to the channel bottom (above the rough areas) (Stoesser, T. & al., 2015). The influence of secondary flows on the evolution of the turbulent shear stress is well demonstrated by the momentum balance. The u’w’ nonlinearity is indeed flowing developed non-parallel, a consequence of the transport amount of longitudinal movement by the secondary flows, as shown by the integration over z of the equation: In our case, the maximum of turbulent kinetic energy (fig.7) is achieved in the fluctuation zone (which is located above the rough area and below the free surface) because the velocity decreases and increases agitation. (Emma, 2010).

Figure 7. Vertical profile of the turbulent kinetic energy

K for the different flow rates (Q1 = 5 l/s; Q2 = 10 l/s;

Q3 = 15 l/s) and the different slopes (a): slope 1%

(b): slope 2%.

4 CONCLUSION

Experiments in a laboratory experimental channel

with a bottom having a uniform roughness were con

ducted. The technique of fast camera has been devel

oped and used for the determination and measurement

of the velocity components. These conducted mea

surements allow the determination of wall parameters

which are fundamental.

It should be noted that one advantage of the bottom

roughness is to slow the vertical velocity of the flow.

In the results found we highlight the presence of a

depression of the maximum velocity below the free

surface. This behavior indicates a retardation of the

flow near the free surface which is a consequence of

the presence of secondary flows in these areas.

Other experiments on bottom with vegetation will

be performed in a large channel at the National Agro nomic Institute ofTunis (INAT), to determine the effect

Study of turbulent flow through large porous media

M. Jouini

National Agronomic Institute of Tunis, University of Carthage, Tunisia

A. Soualmia

Sciences and Technology Water Laboratory (LSTE), Tunisia

G. Debenest

Fluids Mechanics Institute of Toulouse, Group of porous mediums (GEMP), France

L. Masbernat

Fluids Mechanics Institute of Toulouse, France

ABSTRACT: In New Caledonia, the mine tailings are protected from flooding by ensuring the water flow

through riprap. This study refers to high velocity flow in rockfill. It has been defined by the international

firm of engineering consultants MECATER under contract with the Nickel Company of ERAMET group in

New Caledonia (SLN), to be carried at the National Agronomic Institute of Tunis (INA Tunis) in collaboration

with the Fluids Mechanics Institute of Toulouse (IMF Toulouse). In the frame of this study, many experiments

in different types of porous media and under different hydrodynamics conditions were conducted at IMFT. In

this paper we present our simulation results only for homogenous porous media made by spherical particle

with mean diameter d = 1 cm, It is observed that the porous media presented a variation of the permeability in

different flow regimes. The main aim of this study is to restore the permeability variation curve as a function

of the Reynolds number and simulate the experimental

results by Forchheimer, Ergun and Barree and Conway relations. A comparative analysis between these equations was presented.

1 INTRODUCTION

Turbulent flows in highly permeable environments such as blade breezes in the protection of beaches, barren rock storage…, make it a very interesting topic of research recently. A literature review showed that Darcy relationship is no longer valid for this type of flow (Soualmia et al. 2015; Cyprien 2012). Other researchers like Ergun, Forchheimer, Barree and Conway … have developed models to calculate the pressure drop concerning the inertial forces by integrating a second term on V 2 in Darcy model (Barree & Conway 2004; Sano & Kuroiwa 2009).

Each relation depends on physical parameters of the porous media such as permeability, porosity, particle shape…. But it has been shown that the permeability of the studied media is the most important parame ter, since it is the most affected by the flow regime (Wang et al. 1998; Yi et al. 2013). The permeabil ity depends so on Reynolds number; an experimental and theoretical study of this factor was carried out to restore the curve of permeability variation as a function of the Reynolds number. To do so various experiments were carried out with balls and homogeneous stones with a mean diameter d = 1 cm. 2 LITERATURE REVIEW Since Darcy's law is no longer valid for transient and turbulent flow in porous media, many authors proposed different relations of head loss calculation, each one depends on their own experimental results. 2.1 Forchheimer's model (1901) Forchheimer's model is among the first models proposed to calculate the pressure loss when the inertial effects are no longer negligible. It expresses the hydraulic gradient as the sum of two terms, one term is proportional to the speed to simulate Darciens flows, the other one is proportional to the square of the speed (in the case of turbulent flows with dominance of inertial forces) (Betao et al. 2012, Barree & Conway 2004): Where c is a constant for a given structure, V is the fluid velocity, k is the apparent permeability, ν is the dynamic viscosity, and g is the gravity acceleration.

If we express the length scaled depending on the

intrinsic permeability by setting:

n is the medium porosity.

Equation (1) is written as:

J is the head loss.

The friction coefficient c f introduced in the expres

sion of the hydraulic gradient is thus given by:

If the pore Reynolds number is large (Re p), the flow

is dominated by inertial effects and can even be fully

turbulent, and the friction coefficient becomes inde

pendent of the Reynolds number, with c f = nc, but

varies according to the porous structure.

Forchheimer equation has been the subject of

numerous and theoretical studies in the case of porous

medium with periodic structure in which the direct

flow simulation allows the calculation of the parame ters of the pressure drop relationship.

Hydraulic buried wicks are relied on a more empir ical approach to the formulation of the loss, requiring experiences especially in highly complex structure riprap in circles.

To understand the phenomenon, it is necessary to study the medium permeability evolution as a func tion of the Reynolds number. To establish the curve of the permeability change depending on the Reynolds number, a pereametre which is described below was constructed to develop and interpret this curve.

2.2 The Ergun model (1952)

Ergun in his model, expressed the pressure drop depending on the physical characteristics of the porous medium, and proposed the following relationship (Dukhan et al. 2014):

Where, L is the length of the porous medium, ε is the porosity, d is the particle diameter, μ is the dynamic viscosity, and V is the average velocity.

The Ergun relationship is like the Forchheimer one, it expresses the pressure loss as a function of two terms, the first one depends on V taking into account the viscosity and the second one depends on V 2 taking into account the inertial forces.

The Ergun model takes into account the laminar and turbulent flow terms. The difference between the

Forchheimer model and the Ergun one, lies in the expression of the response of the physical characteristics of the medium analysis, in fact Ergun uses porosity and constants that depend on experimental conditions, while Forchheimer uses its own parameter F, and porous medium permeability. 2.3 Barree and Conway model (2004) A general dimentionless model for head loss was proposed by Barree and Conway which normaly cover the totality of velocity flow ranges from darcy flow to fully turbulent flow. For very heigh velocity flow the model describe a constant permeability plateau, otherwise, it will converge for Forcheimer model. In their model, Barree and Conway expressed the apparent permeability as function of Reynold number by the following expression: Where K mr is the minimum permeability divided by the Darcy permeability (K d), as expressed below V: the average velocity in the section, T: Barree and Conway constant expressed as the inverse of a length. In their relation, Reynold number was expressed as function of T which has the dimension of a length inverse [1/L]. This parameter can depends on porous media's mean diameter d. It was expressed as follow: They concluded that more interest must be given to study this parameter. And they determined it with non linear regression in their study. The general proposed Barree and Conway model is expressed as: 3 EXPERIMENTAL RESULTS 3.1 Experimental setup The experiment set up is a cylindrical, permeameter it has a circular section with diameter of 67 cm and

Figure 1. First IMFT experimental set up.

length L = 65 cm powered down by a fan provid

ing a maximum flow rate of Q = 2000 m 3 /hr. This

fan is attached to a divergent cone to ensure the air

distribution throughout the porous medium.

The studied porous media is formed by homo

geneous glass beads which mean diameter d = 1 cm

arranged over a length of 64 cm in the permeameter.

The porosity of this medium was measured, n = 0.402.

The measurement principle is to modify the air flow rate injected, and to measure the pressure difference between the inlet and the outlet of the porous media. To reach very high velocity, we realized the same experiments with the same fan in another permeameter with lower diameter D = 10 cm so we can obtain fully turbulent flow in the studied porous media. Flow velocity was determined using a hot wire manometer as we can see in the figure 2.

3.2 Determination of apparent permeability K_{app}

K_{app} is determined by Darcy's law as following:

V is the flow velocity, Δp is the pressure head, μ is the dynamic viscosity of water and Δx is the wick length.

The following curve of variation of the permeability as a function of Reynolds number in homogenous porous media was obtained.

The figure above shows that the Reynolds number for these experiments varied between 28 and 1100, it is then not in the Darcy flow regime. Hence the variation in the permeability as a function of Reynolds number. In fact the medium permeability decreases with the increasing of Reynolds number.

We obtained two different flow, transient flow for Reynold number $10 < Re < 100$, and turbulent flow for Reynold number under 100.

3.3 Simulation results with the Forchheimer, Ergun and Barree & Conway models

In the literature in addition to the relations of Forch

heimer and barree Conway (Barree & Conway 2004), Figure 2. Second used permeameter and hot wire manometer. Figure 3. Experimental results of the permeability changes as a function of the Reynolds number. there are also Ergun model (Sano et al. 2009; Dukhan et al. 2014) who proposed different head loss models in porous media. The permeability variation is plotted as a function of Reynolds number with these models (Fig 3). Permeability is explained in the Forchheimer model as given bellow:

Figure 4. The apparent permeability variation with the

Reynolds number by studied models.

Mean Ergun Forchheimer Barree and Conway

diameter

d (cm) A B K d F k d K mr T

1 5E-2 112 7.11E-6 0.027 7.11E-6 0.002 30.75

When it is explained by Ergun's model as:

A and B are Ergun's constant.

Apparent permeability given by Barree and Conway

model was given by equation (6).

Different parameters for each relation were defined

before.

The figure above shows a lag between the experi

mental results and those obtained by studied models, in

fact it is shown that, Ergun and Forchheimer models

gives the same results where as Barree and Conway

model gives the nearest simulation results, we can

observe the permeability plateau for Reynolds number

bigger then 800.

The table above presents different obtained values

for each relation.

Each parameter of each relation was determined

basing on our experimental results which explain

the difference between our values and the letterature

values.

Actually we are realizing experiments using a

porous media made by pebbles whish mean diameter

is d = 15 cm in another permeameter. A comparative

Vorticity fluxes on the wake of cylinders within random arrays

A.M. Ricardo & R.M.L. Ferreira

CEris, Instituto Superior Técnico, ULisboa, Portugal

ABSTRACT: Flow around multiple cylinders in squared or staggered configurations have received consid

erable attention but random distributions of cylinders are less studied. The main objective is the study of the

vorticity fluxes in a flow where turbulence is generated by randomly placed cylinders. To achieve this goal a

flow around an isolated cylinder and a flow within a random array of cylinders were experimentally tested. The

experimental databases were acquired with a 2D Particle Image Velocimetry system (PIV) with a spatial resolu

tion that allows the computation of vorticity fields and vorticity fluxes. Results show an accentuated decrease of

the longitudinal vorticity flux in the wake of cylinders within dense arrays, confirming the existence of vorticity

cancellation mechanisms. The strong longitudinal vorticity flux reduction in dense patches is not accompanied

by an increase of the lateral flux of vorticity across the symmetry axis of the cylinders.

1 INTRODUCTION

Flow around circular cylinders is a classical prob lem of fluid mechanics due to its common occurrence in many applications. The flow around an isolated cylinder has been extensively studied achieving impor tant advances in the understanding of the boundary layer, the separating shear layer, the wake and the vortex dynamics (Williamson 1996, Norberg 2003, Parnaudeau et al. 2008, Ranjan and Menon 2015). The vortex shedding regimes are, nowadays, well defined to a considerably large range of Reynolds number (Roshko 1993, Williamson 1996).

Flow around multiple cylinders have also received considerable attention, several studies with two (Kiya et al. 1980, Sumner 2010) and three cylinders (Zhang and Zhou 2001) have been presented. Cases with many cylinders have mainly considered squared or staggered configurations (Lam and Lo 1992, Kim and Stoesser 2011, Nicolle and Eames 2011, Chang and Constanti nescu 2015).The bridge between the actual knowledge for few cylinders in staggered configuration and the global understanding aimed to a random array is still a challenge as configurations with random distribution of cylinders are less studied (Tanino and Nepf 2008, Tanino and Nepf 2009, Ricardo et al. 2014).

Ricardo et al. (2014) presented a spatial characteri

zation of the terms on turbulent kinetic energy (TKE)

conservation equation of a flow within a random array

of cylinders.They observed that turbulence production

is dominated by high vorticity wake-type structures.

These are generated through viscous mechanism sim

ilar to those of isolated cylinders and resulting in the

von Kármán vortex street. They showed also that pro

duction and dissipation rates of TKE are not locally

balanced creating complex spatial patterns of turbu

lence transport. The cumulative effect of convection and turbulent transport of TKE is the generation of background turbulence, i.e, non-locally generated turbulence. The interaction between this background turbulence and vortexes shed by cylinders within arrays results in a faster loss of coherence relatively to vortexes shed by an isolated cylinder. Background turbulence affects the wake signature of a cylinder spreads, confining the wake and enhancing vorticity cancellation (Eames et al. 2011). The mechanism by which the shed vorticity is distributed into the near wake as a result of instabilities in the separating shear layers is not well known, mainly due to the lack of resolution in the measured or modelled vorticity fields and the measurement of vorticity fluxes. Ricardo et al. (2014) reported a shorter life for vortexes shed by cylinders within a random array than by an isolated cylinder. This loss of vortex coherence might be related to vorticity cancellation mechanisms (Hunt and Eames 2002, Moulinec et al. 2004, Eames et al. 2011). These mechanisms have been studied for idealized conditions like isolated bluff bodies or arrays with regular arrangements. The effect of converging/diverging streamlines on the vorticity cancellation within the wake of a single cylinder was studied by Hunt and Eames (2002) for laminar and turbulent straining flows. Moulinec et al. (2004) investigated the vorticity cancellation for cylinders in a staggered array which leads to straining flows similar to the former. They related vorticity cancellation with the accelerated diffusion caused by acceleration/deceleration of the flow between the cylinders of the array. Further studies are necessary to understand in detail these mechanisms, in particular for random distributions of bluff bodies. Motivated by the lack of a detailed characterization of the vorticity field in a domain with strong background turbulence, the present work is aimed at

investigating the mechanisms associated to the vor

ticity fluxes in a flow where turbulence is generated

by randomly placed cylinders. The study of vortic

ity fluxes targets quantitative features of the vorticity

cancellation in space and how it is impacted by dif

ferent geometries. To achieve the stated goal two

experimental tests were carried out: i) flow around an

isolated cylinder (test I) and ii) flow within a random

array of cylinders (test A). The experimental databases

were acquired with a 2D Particle Image Velocimetry

system (PIV) with spatial resolution that allows the

computation of vorticity fields and vorticity fluxes.

Throughout this work, a Cartesian reference frame

is considered, where x_i for $i = 1, 2, 3$ correspond to the

streamwise, spanwise, and vertical directions, respec

tively. u_i and ω_i are the corresponding velocity and

vorticity components.

2 EXPERIMENTAL TESTS

The experimental work was carried out in a 12.5 m long

and 0.408 m wide tilting recirculation flume of the

Laboratory of Hydraulics and Environment of Insti tuto Superior Técnico. The flume has glass side walls, enabling flow visualization and laser measurements.

The flume bottom was covered with a layer of 70 % of gravel (mean diameter of $D_{50} = 7.5$ mm) and 30 % of sand ($D_{50} = 0.8$ mm). The flow was controlled by a venetian-blind gate at the flume's outlet.

Two experimental configurations were considered in the present work: an isolated cylinder (Test I) and an array of cylinders (Test A).

In test I, a cylinder with diameter of $d = 1.1$ cm was placed, in the flume centreline, almost at the end of a 3 m long layer of 70 % of gravel ($D_{50} = 7.5$ mm) and 30 % of sand ($D_{50} = 0.8$ mm). Upstream of this reach, at the flume's inlet a 2 m long reach with larger gravel (3 – 5 cm of diameter) was used to accelerate the development of the flow boundary layer. The dis charge during this test was 4.5 l/s and the channel slope was 2.9%.

Figure 1 shows a plan view of the cylinder distri bution of test A, where ≈ 1400 rigid and emergent cylinders of $d = 1.1$ cm were placed in order to create a pattern with seven wavelengths, each 0.5 m long, with varying cylinder areal number-density. The cylinder areal number-density, m, is defined as the local spatial average of the number of cylinders per unit of plane area and it is expressed in stems/m 2 . Each wavelength

of the cylinder distribution in test A comprises:

• a 15 cm long patch with m = 1600 cylinders/m 2

(dense patch P5);

• a 10 cm long transition patch with an aver

age m of 980 cylinders/m 2 , divided into two 5

cm reaches with m = 1200 cylinders/m 2 and m =

800 cylinders/m 2 , respectively from upstream to

downstream (P6);

• a 15 cm long patch of m = 400 cylinders/m 2 (sparse

patch P3); Figure 1. Plan view of the reach were the measurements of test A were carried out. The solid lines aligned with flow direction indicate the location of the vertical planes measured with PIV and the dashed rectangles indicate the location of the horizontal measurements. Table 1. Features of the experimental measurements and flow properties for each test. Test m h U Re p (cylinders/m 2) (m) (m/s) (-) I - 0.070 0.156 1935 A P3 400 0.067 0.085 1166 A P4 980 0.066 0.086 1187 A P5 1600 0.065 0.088 1231 A P6 980 0.064 0.089 1250 • a 10 cm long transition patch with an average m of 980 cylinders/m 2 , divided into two 5 cm reaches with m = 800 cylinders/m 2 and m = 1200 cylinders/m 2 , respectively from upstream to downstream (P4); For each patch, the areal distribution of cylinders was random and in accordance with a uniform distribution. The velocity measurements were performed in enforced narrow regions without cylinders in the spanwise direction ("measuring gaps" identified in figure 1), whose width is equal to the mean intercylinder distance of the upstream reach. See Ricardo et al. (2014) for more details on the experimental set-up of test A. The flow discharge was 2.3 l/s, the channel slope was 0.9% and within the array of vertical cylinders the flow was a quasi-uniform regime. The free surface exhibited an oscillating behaviour, with amplitude smaller than 2.5 mm, without a measurable impact on the flow at intermediate depths. Table 1 presents variables that describe the flow and the experimental set-up of each test, where h is the mean flow depth, U corresponds to the bulk velocity and Re p = Ud/ν is the

Reynolds number based on the cylinder's diameter, d. The kinematic viscosity, ν, depends on the water temperature, which was measured during the experimental tests. Horizontal ($u_1 \times u_2$) and vertical ($u_1 \times u_3$) 2D maps of instantaneous velocities were acquired with

a Particle Image Velocimetry system (PIV) operated at 15 Hz. As solid targets were employed polyamide seeding particles which specific gravity is 1.03 and the diameters range from 30 to 70 μm, with 50 μm of mean. The cut-off frequency of the turbulent signal, for this seeding material, calculated with the theory of Hjemfelt & Mockros (1996) is about 400 Hz. Once the Nyquist frequency of the PIV time series is 7.5 Hz, the seeding particles ensure the quality of the data acquired in the time domain. In the space domain, applying Taylor frozen turbulence hypothesis and con sidering a mean velocity of 0.15 (m/s) and 0.09 (m/s), for test I and A respectively, the cut-off frequency of 400 Hz means that the smallest resolved turbulence scales are $l_c = 0.4$ mm and $l_c = 0.2$ mm, respectively. These turbulent length scales are smaller than the size of the interrogation windows 0.6 mm, therefore the seeding particles also ensure the quality of the data in the space domain.

Each acquired data set consisted in 5000 image couples performing 5 min of consecutive data. For test I, a horizontal plane was measured at x_3/h =

0.6 and five vertical planes were acquired at x_2/d = -1.09; -0.54; -0.18; 0.59; 0.91 (x_2 = 0 corresponding to the cylinder axis). Regarding test A, two horizontal planes at x_3/h = 0.6 and 10 vertical planes were acquired at each measuring gap as shown in figure 1.

3 VORTICITY EQUATION

The vorticity field is crucial in the study of flow phenomena in highly vortical flows such as wake vortices, mainly because it is independent of the reference frame. Furthermore, the spatial resolution of Particle Image Velocimetry (PIV) measurements is an important advantage because it allows computation of spatial differential quantities (Raffel et al. 2007), thus, enabling 2D representation of the vorticity field and discussion of vorticity fluxes. The adequate computation of spatial differential quantities is directly related to the data spatial resolution: higher resolution leads to more accurate derivatives. The required resolution depends on which turbulent structures are targeted and on the methodology employed to detect them. The present study is based, mainly, in time-averaged quantities and therefore the spatial distribution of these quantities is relatively smooth, allowing a resolution larger than that corresponding to Kolmogorov scale at which the secant and tangent segments are similar.

Equation of vorticity conservation for steady flows

is given by (Chassaing 2000, Brown and Roshko 2012)

where ν is the kinematic viscosity of the fluid and

ω_j and u_j are the j th component of the instantaneous

vorticity and velocity field, respectively.

The fluxes of the vertical vorticity are obtained

by integration of equation (1) over a fixed volume followed by the introduction of the Reynolds decomposition and application of Reynolds averaging. The resulting equation is where primes represent the fluctuation relatively to the time-averaged variable and overlines identify timeaveraged variables. The longitudinal flux of $\bar{\omega}_3$ across a line perpendicular to the flow direction, $F_{1\omega 3}$ is given by Note that in the flow studied herein, the time-averaged vertical velocity, $\bar{u}_3$, is nearly zero. The lateral flux of the out-of-plane vorticity $F_{2\omega 3}$ across a line parallel to the flow direction is given by Next section characterizes the vorticity field in both experimental tests. 2D maps of the terms involved in the vorticity fluxes will also be presented. 4 RESULTS Figures 2 and 3 show non-dimensional time-averaged vorticity maps for test I and for patches P3 and P5 of test A. As known in literature, the vorticity distribution on the wake of a single cylinder consists in a quasi-antisymmetric repeating pattern of paired vortexes caused by the unsteady separation of the flow on the cylinder, identifying von Kármán vortex streets. Figure 3 reveals the same antisymmetric pattern on the wake of the cylinders with the array, which has less space to develop in dense patches. Figure 4 shows 2D maps of the terms $\bar{u}_1 \bar{\omega}_3$ and $u'_1 \omega'_3$ normalized by U^2/d for the isolated cylinder test. $\bar{u}_1 \bar{\omega}_3$ has an anti-symmetric distribution relatively to the longitudinal axis of the cylinder. The strongest magnitudes of this term are found in the outside part of the wake where vorticity $\bar{\omega}_3$ presents the highest magnitudes (figure 2). The distribution of the time-averaged correlation of velocity and vorticity fluctuations, $u'_1 \omega'_3$, is also anti-symmetric and consists

Figure 2. Non-dimensional time-averaged vorticity,

$\bar{\omega}_3 / (U/d)$ in the isolated cylinder test. Flow direction is from top to bottom.

Figure 3. Non-dimensional time-averaged vorticity, $\bar{\omega}_3/(U/d)$, in the array test at: a) P3 and b) P5. Flow direction is from top to bottom.

in two strips of opposite signs each side of the cylinder (figure 4b). This term is almost one order of magni tude smaller than the former. Regarding the viscous term, it is at least two orders of magnitude smaller than the other terms. The term $u'_3 \omega'_1$ is not accessi ble from the present database, however longitudinal vorticity fluctuation are expected to be smaller than vertical vorticity fluctuation. Also u'_3 is smaller than u'_1, hence, this term was considered negligible.

For test A, figures 5 and 6 present maps of $\bar{u}_1 \bar{\omega}_3$ and $u'_1 \omega'_3$, respectively, normalized by U^2/d. The spatial distribution of these two terms involved in the lon gitudinal vorticity flux on the wake of the cylinders within the array are characterized by an anti-symmetric distributions relatively to the longitudinal axis of the cylinder, similarly to that of the isolated cylinder. However, in dense patches like P5 (figures 5b and 6b), the wake of each cylinder is forced to narrow giving the visual impression of compression of the flow variables by the lateral and downstream neigh bours. In particular, the well defined pattern of $u'_1 \omega'$

observed in the wake of the isolated cylinder (fig

ure 4b) and of the cylinders in P3 (figure 6a) loses Figure 4. Terms of the longitudinal vorticity flux a) $\bar{u}_1 \bar{\omega}_3 / (U^2/d)$ and b) $u'_1 \omega'_3 / (U^2/d)$ .Flow direction is from top to bottom. Figure 5. Non-dimensional term $\bar{u}_1 \bar{\omega}_3 / (U^2/d)$ of the longitudinal vorticity flux in the array test at: a) P3; b) P5. Flow direction is from top to bottom. coherence in dense patches (figure 6b). This might be the result of the wake interaction (small inter-cylinder distance) combined with the strong background turbulence (Moulinec et al. 2004). Furthermore, the background turbulence within the array might also explain the increase of the magnitude of $u'_1 \omega'_3 / (U^2/d)$ in test A relatively to the magnitudes observed in the wake of the isolated cylinder. Both sparse and dense patches reveal larger magnitudes (figure 6). The spatial distribution of the vorticity and the terms of its longitudinal flux suggests that, associated to the narrowing of the near wake of the cylinders within

Figure 6. Non-dimensional term $u'_1 \omega'_3 / (U^2/d)$ of the lon

gitudinal vorticity flux in the array test at: a) P3; b) P5. Flow

direction is from top to bottom.

the array, there is a decrease of flux of vorticity. Fig

ure 7 shows the decay of the longitudinal vorticity

flux across the line $x_1 = x^{(i)}$ between the cylinder axis,

$x_2 = 0$, and $x_2 = d$ for the cylinder of test I and two

cylinders in each measuring gap of test A. Generally,

the decrease of the vorticity flux is more pronounced

in the near wake of the cylinders within patches with

larger cylinder areal-number densities (P4 and P5). An

exception is observed for one cylinder in P3 (centred

at $x_2 = 0.137$ cm) which registers a decrease in the vorticity flux similar to that of the cylinders in the dense patch (P5). The TKE magnitude in the wake of this cylinder was observed to be higher than in other cylinders of the same measuring gap, being closer to cylinders in denser patches than to the isolated cylinder of test I, both in terms of magnitude and lateral gradients.

The rate of decrease of $F_{1\omega 3}(x_1, 0 < x_2/d < 1)$ of the cylinders in P3 and P6 (sparser patches) is similar to that observed in the isolated cylinder. For these cylinders (without or with few neighbours) within approximately one diameter from the cylinder base the vorticity flux is about 80% of the flux at $x^{(0)}$ while for the cylinders in denser patches (P4 and P5) the flux at $x_1 = d$ reduces to 60% of the flux at $x^{(0)}$. For the cases studied herein, the decrease of $F_{1\omega 3}(x_1, 0 < x_2/d < 1)$ within one diameter downstream is smaller than the 50% reduction reported by (Brown and Roshko 2012).

Regarding the lateral vorticity flux, the present database allows 2D horizontal maps of the terms $\bar{u}_2 \bar{\omega}_3$, $u'_2 \omega'_3$ and $\nu \frac{\partial \bar{\omega}_3}{\partial x_2}$, that revealed stronger valued on the wake and on the vortex street of each cylinder, as was observed in the terms of $F_{1\omega 3}$. The term $u'_3 \omega'_2$ is computed with data from vertical planes.

Figure 8 presents maps of the terms $\bar{u}_2 \bar{\omega}_3$ and $u'_2 \omega'_3$

normalized by U^2/d for test I.

The term $\bar{u}_2 \bar{\omega}_3$ is symmetric, presenting strong mag

nitudes on vortex street and decreasing fast to zero out

of this region. It consists in a large lobe of positive
Figure 7. Ratio of longitudinal vorticity flux at x (i) and the initial point, x (0) , which is the closest point to the cylinder available in the experimental database. Two cylinders were considered for each measuring gap. Figure 8. Non-dimensional terms of the lateral vorticity flux a)$\bar{u}_2 \bar{\omega}_3 / (U^2/d)$ and b) $u'_2 \omega'_3 / (U^2/d)$. Flow direction is from top to bottom. values, starting at about 0.5d and extending further downstream, preceded by a small lobe of strong negative values (figure 8a). Also symmetric relatively to the cylinder's axis, the term $u'_1 \omega'_3$ is positive on the cylinder's wake and has a strip of negative values on the vortex street (figure 8b). $\bar{u}_2 \bar{\omega}_3$ is larger than $u'_2 \omega'_3$, but of the same order of magnitude. The viscous term also presents the largest values on the cylinders wake, but it is at least one order of magnitude smaller than the other terms. Term $u'_3 \omega'_2$ being obtained from measurements in the vertical plane does not allow a spatial characterization with the same detail as in 2D horizontal maps. Nevertheless, it was possible to conclude that this term is larger in the vortex street region than in

Figure 9. Non-dimensional term $\bar{u}_2 \bar{\omega}_3 / (U^2/d)$ of the lat

eral vorticity flux in the array test at: a) P3; b) P5. Flow

direction is from top to bottom.

the wake centre. Furthermore, its magnitude is smaller

than that of $u'_2 \omega'_3$.

Figures 9 and 10 present maps of $\bar{u}_2 \bar{\omega}_3$ and $u'_2 \omega'_3$

normalized by U^2/d for two measuring gaps of test A

(P3 and P5). As observed in the terms of the longitu dinal flux, the spatial distribution of these two terms involved in the lateral vorticity flux on the wake of the cylinders within the array are similar to that of the isolated cylinder. Figures 9 and 10 also show that the magnitude of $\bar{u}_2 \bar{\omega}_3 / (U^2/d)$ and $u'_2 \omega'_3 / (U^2/d)$ is slightly larger in the wake of cylinders of test A than in test I. This reinforces the role of the back ground turbulence:the interaction of the vortexes shed by a given cylinder with the non-locally generated turbulence seems to increase the correlation between velocity and vorticity fields relatively to that of an isolated cylinder. $u'_3 \omega'_2$ on the wake of the cylinders in test A also reveals magnitudes smaller than the term $u'_2 \omega'_3$. Figure 11 illustrates 2D maps of this term in the wake of a cylinder of of the measuring gap P4. $u'_3 \omega'_2$ presents a symmetric distribution with negative values in the center of the near wake (figure 11a) and positive on the outer part of the wake (figure 11a).

Figure 12 presents the lateral flux of vorticity across the plane $x_2 = 0$ (cylinder axis) as function of the normalized distance downstream of the cylin der base, x/d, in the near wake of cylinders of tests I and A. The dominant term of the lateral flux of vorticity across the cylinder symmetry axis,

$F_{2\omega_3}(x_1, x_2 = 0)$, is the time-averaged correlation of

lateral velocity fluctuations and vertical vorticity fluc

tuation, $v'\omega'_z$ and the remaining terms are vanishingly

small, so that $F_{2\omega_3}(x_1, x_2 = 0) \approx u'_2\omega'_3$. The shape of

$F_{2\omega_3}(x_1, x_2 = 0)$ is similar for all the cylinders and is in

accordance with that presented by Brown and Roshko

(2012) from a Large Eddy Simulation (LES) of an

isolated cylinder. Figure 10. Non-dimensional term $u'_2\omega'_3/(U^2/d)$ of the lateral vorticity flux in the array test at: a) P3; b) P5. Flow direction is from top to bottom. Figure 11. Non-dimensional term $u'_3\omega'_2/(U^2/d)$ of the lateral vorticity flux in the measuring gap P4 of test A at: a) $x_2 = 0.214$ cm (cylinder axis); b) $x_2 = 0.224$ cm. Flow direction is from left to right. Except at the close vicinity of the cylinder base where it may lightly decrease, the lateral flux of vorticity is expected as an increasing function in the near wake and becoming almost constant for $x/d > 4$ (Brown and Roshko 2012). Figure 12 shows that $F_{2\omega_3}(x_1, x_2 = 0)$ for most of the cylinders within the array starts increasing closer to the cylinder base than in the isolated cylinder but the rate of growth also decreases closer. At $x/d = 2$ the lateral flux of vorticity measured in the wake of the single cylinder of test I is larger than in almost all cylinder of test A. Although the differences between the isolated cylinder and the cylinders within the array are relatively small, the vorticity flux across the symmetry axis of a cylinder seems to decrease with the proximity to neighbouring cylinders, however it depends on the particular local distribution of the neighbours and its impact on the local instabilities in the shear layers. The present discussion shows that the more accentuated reduction of the longitudinal vorticity flux in

Figure 12. Lateral vorticity flux $F_{2\omega_3}(x, y = 0)$ for the iso

lated cylinder of test I and two cylinders of each measuring gap of test A.

the wakes of cylinders in denser patches is not accom panied by an increase of the lateral flux of vorticity across the symmetry axis of the cylinders.

5 CONCLUSIONS

The present work employed experimental databases acquired with a 2D PIV in a turbulent flow within an array of cylinders with varying cylinder areal number density. A flow around an isolated cylinder was also explored. The main objective was to investigate the role of the vorticity fluxes in the wake of the cylinders. Due to the nature of the experimental database, this study allowed a detailed characterization of the most relevant vorticity component, the vertical vorticity, and its fluxes. The decrease of the longitudinal vorticity flux, along the streamwise direction, revealed to be more accentuated in the wake of cylinders within dense arrays. This faster decrease of the longitudinal vortic ity flux observed in the wake of a cylinder with close neighbours could generate expectations to an increase of the lateral vorticity flux relatively to the isolated cylinder, however, the vorticity fluxes across $y = 0$ are similar or even smaller in the wake of cylinders within the array.

The conclusion that the pronounced reduction of the longitudinal vorticity flux in the wakes of cylinders in

dense patches is not accompanied by an increase of the

lateral flux of vorticity across the symmetry axis of the

cylinders confirms the existence of vorticity cancel

lation mechanisms. The vorticity cancellation within

random arrays of cylinders might be caused by the

mechanisms associated to straining flows explained

by (Hunt and Eames 2002, Moulinec et al. 2004) but

also by turbulent transport.Another mechanism of vor

ticity cancellation may be the transformation of the

Ranjan, R. & S. Menon (2015). On the application of the two

level large-eddy simulation method to turbulent free-shear

and wake flows. Journal of Turbulence 16(2), 136–166.

Ricardo, A. M., S. Di Carlo, M. J. Franca, A. J. Schleiss, P. M.

Sanches, & R. M. Ferreira (2014). Vortex interaction in

patches of randomly placed emergent cylinders. In S. et al

(Ed.), Proceedings of River Flow 2014, pp. 63–69.

Ricardo, A. M., K. Koll, M. J. Franca, A. J. Schleiss, & R. M.

Ferreira (2014).The terms of turbulent kinetic energy bud

get within random arrays of emergent cylinders. Water

Resources Research 50(5), 4131–4148.

Roshko,A. (1993). Perspectives on bluff body aerodynamics.

Journal of Wind Engineering and Industrial Aerodynam

ics 49(1), 79–100. Sumner, D. (2010). Two circular cylinders in cross-flow: A review. Journal of Fluids and Structures 26(6), 849–899. Tanino,Y. & H. M. Nepf (2008). Lateral dispersion in random cylinder arrays at high reynolds number. Journal of Fluid Mechanics 600, 339–371. Tanino, Y. & H. M. Nepf (2009). Laboratory investigation of lateral dispersion within dense arrays of randomly distributed cylinders at transitional reynolds number. Physics of Fluids 21(4), 046603. Williamson, C. H. (1996). Vortex dynamics in the cylinder wake.Annual Review of Fluid Mechanics 28(1), 477–539. Zhang, H. J. & Y. Zhou (2001). Effect of unequal cylinder spacing on vortex streets behind three side-by-side cylinders. Physics of Fluids 13(12), 3675–3686. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Effect of downstream channel slope on numerical modelling of dam

break induced flows

A.E. Dinçer, Z. Bozkuş, & A. N. Şahin

Middle East Technical University, Ankara, Turkey

ABSTRACT: In the present study, the numerical simulations of dam break induced flows are performed by

using various models. The numerical models used in the study are laminar, large eddy simulation (LES) and

Reynolds-Averaged Navier Stokes (RANS) equations with k-ε turbulence model. In addition a recently developed

Smoothed Particle Hydrodynamics (SPH) code is also used to simulate dam break problem. For the validation

of the numerical approaches, a recently published experimental study is used. In the experimental study, an

idealized dam break problem in a sloped channel is investigated. It is found that although other numerical

approaches give similar velocity profiles with the experiments, RANS equations with k-ε turbulence model

gives underestimated results. The same problem is simulated by assuming that the downstream channel is not

inclined, i.e. horizontal. It is observed that when the

channel is not inclined, k-ε turbulence model gives closer results to other numerical models. Therefore, it can be concluded that for dam break induced flows, slope of the channel may adversely affect the accuracy of k-ε turbulence model. However, more experimental and numerical data is needed to validate this conclusion.

Keywords: Dam break, turbulence modelling, Large Eddy Simulation, LES, k-ε turbulence model, laminar flow

1 INTRODUCTION

Turbulence modelling has been, and to this date still is, a significant research subject. The problems in numerical modelling arise due to complex nature of turbulence. The earliest attempts in turbulence mod elling started with the introduction of eddy concept by Boussinesq (1877). Later, mixing-length model con cept known as zero-equation model was introduced by Prandtl (1925). Kolmogorov (1942) introduced k-ω model which is the first complete turbulence model. One-equation model which includes the flow history was also developed by Prandtl (1945). In order to solve Reynold's stress, Boussinesq approximation in turbulence models was developed by Rotta (1951). Boussinesq approximation is a second order closure model. After this point, with the increase of computer power, the progress of these models accelerated. In mixing-length model, a viscous damping correction, still in use in turbulence models, was developed byVan

Driest (1956). Eddy viscosity and mixing-length con

cept was refined by Cebeci and Smith (1974). Launder

and Spalding (1972) developed k-ε turbulence model

which is the most popular two-equation turbulence

model.

The dam break problem is also simulated with

Smoothed Particle Hydrodynamics (SPH) which was

developed by Monaghan to model astrophysical prob

lems. Later, Monaghan (1994) applied SPH to open

channel flows. In the SPH, fluid is simulated with small particles which show all the characteristics of the fluid. In SPH, the fluid is usually assumed as slightly compressible instead of truly incompressible to avoid solving Poisson's equations (Colicchio et al., 2002). For a more detailed information about SPH, one can investigate the studies of Liu and Liu (2003) or other studies of the authors. In this study, a dam break problem is simulated numerically. A laminar flow model and two turbulence models, k-ε model and Large Eddy Simulation (LES) model, are used in simulations. In order to verify the numerical models, the experimental data of LaRocque et al. (2013) is used. The layout of the paper is as follows: First the numerical models are defined, then the experimental and numerical setups are given, later the results are presented and finally conclusions are drawn. 2 NUMERICAL MODELS No matter turbulent or not, all fluid motions follow dynamical equation of fluids. A valid description of laminar and turbulent flows is given with Navier-Stokes equations combined with the continuity equations:

where x_i and x_j are the Cartesian coordinate com

ponents, g is the gravitational acceleration, ρ is the

density of the fluid, u is the velocity component, μ is

the dynamic viscosity, P is the pressure, and $-\rho u'_i u'_j$ is

the Reynolds stress which represents the effect of tur bulence. The equations of k-ε turbulence model can

be shown as:

where P_b is the buoyancy force and μ_t is the turbu

lent viscosity. For more detailed information about

equations, the works of Wilcox (1998) can be studied.

LES equations are shown as:

whereσ_{ij} is the stress tensor due to molecular viscosity.

In the simulations, for the solutions of laminar flow,

k-ε turbulence model and LES model OpenFOAM,

an open source computational fluid dynamics (CFD)

software, is used.

The governing SPH equations can be stated as: where i and j show the particle of interest and the neighboring particle, m is the mass, g is the gravitational acceleration, P is the pressure, W is the kernel function which shows the effect of the neighboring particles to the particle of interest and π_{ij} is the socalled Monaghan-type artificial viscosity. In the study cubic spline kernel is used. In SPH code, XSPH technique is used. XSPH forces the particles to move with a closer velocity to the average velocity of the neighboring particles (Monaghan 1989; 1992). XSPH can be stated as: In the code, water is assumed as slightly compressible and an equation of state is used to apply this assumption. According to equation of state, a slight change in density can cause a large variation in pressure. where P_o is the initial pressure. Boundary particles are placed at the boundaries to define walls or obstacles. The boundary particles exert repulsive force to the water particles and the magnitude of the force increases while the spacing between water and boundary particles decreases. This force can be calculated as: where r_0 is the initial spacing between water particles and D is the problem dependent parameter and taken in the same scale with the square of the largest velocity (Liu and Liu, 2003). The boundary particles are placed with half of the initial spacing of water particles to provide a solid barrier.When $r > r_0$ the force is taken as zero. To integrate SPH equations leap-frog algorithm is used and time stepping is controlled with CourantFriedrichs-Lewy (CFL) condition (Anderson, 1995). 3 EXPERIMENTAL AND NUMERICAL SETUPS The experimental setup of LaRocque et.al.(2013) is given in Figure 1. They used a channel having 7.31 m length, 0.18 m width and 0.42 m depth. The bottom slope of the channel was 0.93%. 3.37 m from the upstream end, there was a wooden gate. This wooden gate was lifted suddenly in order to simulate instantaneous dam failure. The removal time of the gate

Figure 1. Experimental Setup of LaRocque et al. (2013).

was recorded as 0.21 s for an initial reservoir head

of 0.35 m. Lauber and Hager (1998) suggested that

the gate opening can be considered as instantaneous if

$T_l \leq \sqrt{2D/g}$.According to this formula, the gate open

ings in the experiments were instantaneous. In order

to measure water depths Baumer ultrasonic distance

measuring sensors were used. In addition, velocities

were recorded with an ultrasonic velocity profiler

(UVP) Velocities are recorded at a very fast rate with

UVPs. For the measurement of water surfaces, the

probes were placed at −0.3, −0.5, −0.7, −0.8, −0.9,

−1.1, and −1.5 m upstream, and 0.2, 0.4, 0.6, 0.8, and

1.0 m downstream of the gate. For the velocity mea

surements, probes were placed in a horizontal position,

0.045 m above the channel bed. In this study, only the

simulations of most upstream and most downstream

measurement points from the wooden gate are pre

sented. In addition, the simulation results for water

surfaces are not presented. For the results of these sim

ulations one can see the other studies of the authors about this subject.

The main scope of this study is to investigate the effect of downstream channel. First, the channel with a 0.93% slope was simulated. Then, the channel was assumed as horizontal and the simulations were repeated.

For the numerical simulation of laminar flow, RANS equations with k-ε turbulence model and LES model, the parameters are given in Table 1. In the simulations, the wall boundary with no-slip condition was used for the upstream end of the reservoir and for the bottom of the channel. In addition, symmetry boundary condition was used for the top of the computational area, and outflow boundary condition was used for the downstream end of the reservoir.

The computations are stopped after 2.5 s. The initial water height and length were defined according to the experimental values (h_0 = 0.25 and 0.30). The time step was defined according to Courant Friedrichs-Lewy (CFL) condition. In the k-ε turbulence model, k = 0.01 and in the LES model C_ε= 1.048, C_k = 0.094.

In the SPH simulations, the initial distance between water particles is taken as 0.01 m. On the other hand,

Table 1. Parameters used in mesh-based methods. Grid size Grid size Grid size in x in y in z Simulation direction

direction direction time Model (m) (m) (m) (s) Lam. 0.005 0.005 0.01 2.5 RANS 0.005 0.005 0.01 2.5 LES 0.0015 0.0015 0.01 2.5 Table 2. Relative difference between numerical and experimental results (in %). % relative differences between X ′ SPH-exp. lam-exp. RANS-exp. LES-exp. 2.00 4.02 4.12 8.29 4.57 4.00 5.99 4.44 8.00 5.08 the boundary particles are placed with half of the initial spacing between water particles, i.e. 0.005 m. The mass of the particle is calculated by multiplying the initial spacing between two particles and density of the particle. By knowing initial pressure, which is hydrostatic pressure, the densities of the particles are calculated from equation (10). 10110 water particles and 1861 boundary particles are used in the simulations. The smoothing length is taken as 0.01 m. Time step is 2.5 e-5 s and the simulation continues for 4 s. 4 RESULTS To present the results, dimensionless parameters, X ′ = x/h 0 , X ′′ = x/(t(gh 0) 0.5), V ′ = v/(gh 0) 0.5 and T = t(g/d) 0.5 , were used where X ′′ shows the dimensionless distance from the probe used in the experiments.The probe was placed 0.045 m above the bottom of the flume.

Figure 2. Velocity Profile with a slope of 0.93% at X ′ = 2.00.

Figure 3. Velocity Profile with a slope of 0.93% at X ′ = 4.00.

The numerical simulations compared with the

experimental data for X ′ = 2.00 and X ′ = 4.00 which

are the most upstream and most downstream points

from the gate are shown in Figures 2 and 3 respec

tively. In the figures, horizontal velocity values at the downstream side of the gate for an initial reservoir head of 0.30 m are given for laminar flow, SPH model, RANS and LES results and experimental data for T = 11.44. As can be seen, all the simulations are in agreement with the experimental data. However,

Figure 4. Velocity profile when the channel is horizontal at X ′ = 2.00.

Figure 5. Velocity profile when the channel is horizontal at X ′ = 4.00.

RANS equations with k-ε turbulence model give the furthest results. The difference between numerical and experimental results is given in Table 2. As can be seen from the table, k-ε turbulence model underestimates the results nearly 8%, although other models deviate from experimental data between 4% and 6%. In fact, the results from k-ε turbulence model are also successful (8% difference is not too much for a numerical simulation) but the authors wanted to study why such a result occurred. One of the reasons is deemed to be channel slope. In order to investigate the effect of channel slope, the simulations are done by assuming the channel is horizontal. The results for the same points are shown in Figures 4 and 5. Since there is no avail able experimental data for this case, only the results of numerical simulations can be shown. This time all the simulations including RANS equations with k-ε turbulence model give close results. In fact, the maxi mum difference between two numerical models (LES and laminar flow) is 1.3%. Therefore, it can be con cluded that, the channel slope may adversely affect the correctness of k-ε turbulence model. However, this suggestion should be further investigated.

5 CONCLUSION

In this study, a dam break problem was simulated with a recently developed SPH code, laminar flow, k-ε tur bulence model and LES model. The simulations were verified with the experimental data available in litera ture. It was seen that when there is even a very small channel slope, the results of k-ε turbulence model devi ate from the other simulation results and experimental data. Here it should be noted that this deviation is still fewer than 10% which is an acceptable limit for a numerical simulation. However, when there is no slope, the results of k-ε turbulence model are close to laminar model and SPH model. In fact, there are no suggestions that propose the channel slope may affect the correctness of k-ε turbulence model in literature. Therefore, the authors avoid giving such a conclusion, but continue studying this phenomenon further by con ducting a set of experiments for different bed slopes and comparing the experimental data with the results obtained from k-ε turbulence model.

Anderson, J. D. (1995). “Computational fluid dynamics: the

basics with applications”. Mc Graw-Hill. Boussinesq, J. (1877), “Theorie de l’Ecoulement Tourbillant,” Mem. Presentes par Divers Savants Acad. Sci. Inst. Fr., Vol. 23, pp. 46–50. Cebeci, T. and Bradshaw P., (1988) Physical and Computational Aspects of Convective Heat Transfer, Springer, New York. Colicchio G., Colagrossi A., Greco M. and Landrini M. (2002). “Free surface flow after a dam break”. Ship Technol.Res. 49:95–104. Izvestia Academy of Sciences, USSR; Physics, Vol. 6, Nos. 1 and 2, pp. 56–58. Kolmogorov, A. N. (1942), “Equations of Turbulent Motion of an Incompressible Fluid.” LaRocque, L., Imran, J., and Chaudhry, M. (2013). “Experimental and numerical investigations of two-dimensional dam-break flows.” J. Hydraul. Eng. Lauber, G., and Hager, W. (1998a). “Experiments to dambreak wave: Horizontal channel.” J. Hydraul. Res., 36(3), 291–307. Launder, B. E. and Spalding, D. B. (1972), Mathematical Models of Turbulence, Academic Press, London. Liu, G. R., Liu, M. B., (2003). “Smoothed Particle Hydrodynamics: a Meshfree Particle Method”. World

Scientific Publishing Co. Pte. Ltd., Singapore. Monaghan J. J. (1989), On the problem of penetration in particle methods. Journal of Computational physics, 82:1–15. Monaghan J. J. (1992), Smoothed particle hydrodynamics, Annual Review of Astronomical and Astrophysics, 30:543–574. Monaghan, J. J., (1994). "Simulating free surface flows with SPH". Journal of Computational Physics 110, 399–406. Prandtl, L. (1925), "Uber die ausgebildete Turbulenz," ZAMM, Vol. 5, pp. 136–139. Prandtl, L. (1945), "Uber ein neues Formelsystem fur die ausgebildete Turbulenz," Nacr. Akad. Wiss. Gottingen, Math-Phys. Kl. 1945, pp. 6–19. Rotta, J. C. (1951), "Statistische Theorie nichthomogener Turbulenz," Zeitschrift fur Physik, Vol. 129, pp. 547–572. Van Driest, E. R. (1956), "On Turbulent Flow Near a Wall," Journal of the Aeronautical Sciences, Vol. 23, p. 1007.
Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Analysis of dotation discharge impact at a fishway entrance via numerical 3D CFD simulation

J. Klein & M. Oertel

Hydraulic Engineering Section, Civil Engineering Department, Lübeck University of Applied Sciences, Lübeck, Germany

ABSTRACT: The success of a fishway depends on two criteria: (1) fish passage capability, and (2) fish way detectability. Only if both criteria are fulfilled a fishway can be considered as successful. To design a passable fishway structure defined geometric value limits due to hydraulics and fish physiology must be con sidered. Additionally, the entrance of the fishway must be passable as well as detectable. The present paper deals with a planned fishway in northern Germany. Due to tidal influenced flow parameters, the fishway outlet was designed to be orthogonal to the downstream river system. Hence, negative influences concerning fish entering the structure as well as ship movement can be assumed. Besides passable flow velocities and a minimum of

turbulence at the entrance area, the appearance of a continuous leading flow with velocities larger than 0.2 m/s are

important parameters to guarantee a successful structure. These parameters were taken into account to design

a fish friendly entrance. Furthermore, public authorities gave limitations of maximum mean inlet velocities

because the downstream water body is a small federal waterway with ship traffic where obstruction has to be

avoided. The entrance of the fishway as well as 100 m of the downstream water body is analyzed via numerical

3D CFD simulations. The main aim is to verify the hydraulic flow situation at the inlet zone of the fishway.

Investigations include three different inlet structures and results lead to an optimized design of the fishway's

entrance.

1 INTRODUCTION

Due to the European Water Framework Directive WFD

(2000) all anthropogenic modified waterbodies must

be revert into good ecological conditions. Thereby, the

main criteria to identify a river's patency are func

tional fishstep systems to allow upand downstream

fish migration. Several fishsteps can be used to pro

duce hydraulic flow systems with fish climb capability.

Technical structures (e.g. vertical slot pass) can be

arranged on small footprint areas next to weirs with

hydro-power systems included. Nature-like solutions

ought to provide a natural flow situation and an attrac

tive structure within the environment.These can be e.g.

crossbar block ramps (Oertel & Schlenkhoff 2012) or ramps with interlocked blocks.

The present paper deals with a special kind of natural structure – a brush-furnished fishway – and analyzes hydraulic impacts within the downstream river system via numerical 3D CFD simulations.

2 SUBJECT OF RESEARCH

The study area is located in the north of Germany. The project deals with a fishway connecting two heavily modified waterbodies which were separated for more than a hundred years (Coleman 1926, Gripp 1982). The river system of Trave and Wakenitz are shown in Fig. 1. Waterbody 2 (Wakenitz) is strictly impounded by a weir yielding a fixed water level. Fig. 1 shows a channel at that's end discharge through a culvert system is promoted. Even so, an investigation of the river system showed the possibility to convert the flow system into good ecological conditions (L+W Cons. Eng. 2014). Waterbody 1 (Trave) is an estuary of the Baltic Sea and has a tidal range of about 0.2 m in area of investigation. The height difference of both water surface elevations is approximately 4 m with flow direction of the fishway from Wakenitz to the River Trave. For more Information about the river system see Klein & Oertel (2015). The planned fishway was designed as a pool-step system with a total length of 140 m. 24 pools overcome a total difference of water surface elevation of 4 m (L+W Cons. Eng. 2014). The chosen design of the fishway is comparatively new. The so called brushfurnished fishway was developed by Hassinger (2002). Within the brush-furnished fishway pools, are separated by one or more brush-furnished obstacles with openings as a migration corridor. The location of the fishway is shown in Fig. 1. Due to European and national guidelines fishways must be functionally operational 300 days a year. Beyond this period, the fishway operates on minimum load to enable small fish and benthos organisms to

Figure 1. Schematic sketch of flow system.

Table 1. Design discharges of planned fishway.

	FW	DOT	CUL	FRS
	$l s^{-1}$	$l s^{-1}$	$l s^{-1}$	$l s^{-1}$
MLW 30	330	0	350	0
MW 150,Summer	430	690	680	0
MW 300,Winter	600	3600	680	0
MHW 330	600	4000	3500	excess

FW = fishway, DOT = dotation flow rate, CUL = culvert, FRS = flood release structure

migrate. The fishway's flow rate varies between 330 and 600 l/s – but can also reach zero discharge for low water situations. Furthermore the fishway is charged with an additional flow rate at the outlet – the dotation discharge. This dotation is to lead to a better detectability for fish within the downstream river system. Dotation discharges will be guided via a pipe system into the downstream outlet pool.

Wakenitz' flow system offers maximum discharges of 20 $m^3 s^{-1}$, which can be separated into four discharge groups (see Tab. 1, Fig. 2): (1) mean low water level (MLW), (2) and (3) regular operation for 300 days per year (mean water (MW), summer and winter events) – see German design guideline DWA-M 509 (2014) – (4) mean high water (MHW) and higher flood events. It must be considered that discharges above 8.1 $m^3 s^{-1}$ are managed by a flood release structure, located far downstream the fishway.

3 FORMULATION OF THE PROBLEM

Due to the general fishway's position several oper

ational problems can be assumed which need to be

investigated. The fishway's outlet was designed to

be orthogonal to the downstream located Trave to

ensure better detectability for fishes according to con

sulted biologists. Fishes near the investigation area are

more attracted by chemical hormonal parameters than Figure 2. Discharge duration curve of Wakenitz river system, L+W Cons. Eng. (2014). by guiding flow currents as a result of tides (L+W Cons. Eng. 2014). Flow direction will be reversed with tidal influences, hence no migration direction can be indicated by flow direction (L+W Cons. Eng. 2014). Present studies of Katopodis (2015) corroborate that attraction of fishways may depend on biological features more than thought till now. On that account, it is assumed that the chemical and hormonal differences between Trave and Wakenitz induce a migration intention and lead fishes to the fishway coming from downstream and upstream Trave (L+W Cons. Eng. 2014). The fishway's location was provided by the engineers and local authorities assigned and is not the subject of the present paper. Additional dotation flow, charged at the fishway's outlet, is supposed to produce a sufficient volume of water into waterbody 1 (Trave) in contrast to the different chemical hormonal water properties of waterbody 2 (Wakenitz) to stimulate fishes' migration. Supportive, dotation discharges should form a guiding flow to support the structure's detectability. Consequently, high fishway discharge rates of more than 4 $m^3 s^{-1}$ lead to downstream inflow impact at high velocitiy and high turbulence depending on the inflow geometry which results in reduced fish climb capability and negative to dangerous impacts on watercraft movement and their navigability in the downstream river system. To avoid negative influences from the planned fishway, the outlet structure must be investigated in detail. Therefore, three criteria are used for evaluation: (1) fish climb capability, (2) finding the entrance in terms of guiding flow and (3) watercraft impact. 4 NUMERICAL MODEL The present numerical model covers 90 m of waterbody 1 (40 m downstream, 50 m upstream the fishway's inlet) and nine pools of the planned fishway. The river was constructed manually, based on a digital terrain model (resolution 5 m in xand y-direction) and additional cross profiles. The implementation of the fishway is based on the planning documents. More details about the numerical model can be found in Klein (2015).

FLOW-3D (v.11) was used for numerical 3D CFD model simulations; including Volume-of-Fluid method (VOF) and RNG-Turbulence-Model. RNG turbulence model is based on k-ε-model with improved calculation for areas of high pressure gradients at wall (Flow Science Inc 2014). The flow parameters are solved by applying the finite-difference-method on a structured, Cartesian grid. Flow Science Inc (2014) gives additional information. The geometry is embed ded in a number of structured quadratic meshblocks with varying resolution, while the FAVOR™ method is applied. The mesh resolution at the inlet area is 0.1 m (edge lengths of cubic cells). The remaining river geometry is mapped with a cell size of 0.2 m (only one mesh size was computed).And consequently, approximately 4 million active cells are computed.

The present paper analyzes numerical simulation results for a winter event (see Tab. 1, with dotation discharge) concerning flow impacts within the down stream water body for various geometrical solutions (siee also Fig. 3):

1. current planning state,

2. enlargement of outflow pool (Option 1),

3. enlargement of outflow pool and flow guiding wall (Option 2).

5 RESULTS AND DISCUSSION

5.1 Passability – fish climb capability

The passability for fishes is evaluated via maximum passable velocities in the inlet pool and its distribu tion. Relevant fish species are metapotamal species such as pike, brace, perch and bream L+W Cons. Eng. (2014). Maximum passable velocity of the relevant fish species of 1.45 ms −1 should not be exceeded (DWA-M 509 2014). Furthermore it is necessary to ensure the occurrence of resting areas(DWA-M 509 2014)(Adam & Lehmann 2011). Additionally, a diverse velocity distribution is preferred to offer migration corridors for weak species to migrate and large vortex structures should be neglagted because they can lead to loss of orientation (Smith et al. 2014). Figures 5 and 6 show the velocity magnitude at the downstream cross-section of the fishway exactly at the entrance and at a distance of 2 m upstream the fish way. The cross section's location is shown in Fig. 4. YZ-intersection is shown with x-axis as geographical Y-coordinates (Gauss-Krüger Zone 3; GK3) and y-axis as z-coordinates in meters above sea level. With these cross-sections the passability at the inlet can be eval

uated. It follows that the original configuration with

narrow inlet pool results in too high velocities at the

entrance pool (see Fig. 6a). Fishes migrating upstream

cannot overcome high velocities occurring throughout

the entire pool width. This configuration does not pro

vide any migration corridor. The narrow pool leads to

too constricted room for dotation discharge to intermix

with the fishway's discharge and consequently to no Figure 3. Geometry of simulation model. reduction of velocities. Furthermore, it can be considered that high turbulence areas can be found for this planning state. Option 1 and 2 (Figs. 5b/c and 6b/c) clearly show an improved flow situation. By opening and enlarging the inlet pool more space is given to add the dotation discharge. The current enters the fishway with smaller

Figure 4. Location of exported cross-sections, marked in

red.

Figure 5. Cross-section at inlet pool, velocity magnitude.

velocities and mixing within the pool is supported.

Additionally, rotating the dotation profile towards the

river provides a smooth intermixture. Hence, turbu

lence and velocities are reduced. Figures 5b/c and 6b/c

show that high velocities occur only locally. A diverse

velocity profile is given for all relevant fish species

over the pool length; see also Figure 8.

Consequently, enlarging the inlet pool leads to an

improved passability for fishes.

5.2 Detectability – fishway findability

Evaluating the fishway's findability (detectability) is the most complex issue that can be achieved by approximation and simplification of the problem solely. A major factor for detectability is the fishes' behavior, which is not yet fully understood and still the subject of research projects (e. g. Lehmann & Gischkat Figure 6. Cross-section in 2 m distance of inlet pool, velocity magnitude. (2013)). However, certain criteria must be defined in order to evaluate the detectability and passability. According to Adam & Lehmann (2011) upstream migration is stimulated when flow velocity exceeds 0.2 ms −1 . Hence, a migration corridor with a velocity of 0.2 ms −1 should be achieved. The matter for evaluation is the appearance of the migration corridor in terms of intermixture, shape (length, width) and velocities. Figure 7 shows the guiding flow occurring.Velocity distributions exceeding guiding flow limiting velocities of 0.2 ms −1 are illustrated. Flow velocities below 0.2 ms −1 are neglected in figure 7. Simulation results of planning state show a heavily developed transversal current which flows almost orthogonally to the main channel. Hence, a stimulation of migration can be assumed. Since the migration direction is crosswise, the orthogonal direction of the resulting guiding flow could be problematic – the fishes' reaction to this phenomena is uncertain. For option 1 the intermixture of the inflowing water is much stronger.The transversal current forms a guiding flow at the fairway. The guiding flow is deflected and, thus, directed in main flow direction. The guiding flow may be detectable more than 40 m downstream the fishway. Simulation with an installed wall (option 2) shows a deflection and constriction of the guiding flow. Hence, the guiding flow develops at the right embankment at a width of 5 m. The length exceeds the numerical model domain.

Accordingly, a guiding flow, as defined previously, will be developed for options 1 and 2. Again, the planning state solution will include several disadvan tages. Immense dotation discharges lead to an almost orthogonal current which breaks through the main flow direction. Option 1 shows a distinct leading flow over 80 % of the main channel width. The developed guid ing flow is directed in flow direction and located in the fairway. As expected, an installed wall leads to sepa ration of fairway and transversal current and guiding flow is located at the right embankment.

In conclusion, detectability was improved with option 1 and 2. Further investigations must clarify if the separation of main channel and embankment leads to a reduced detectability concerning the migration corridor in the natural river.

5.3 Watercraft impact

The safety for ship movement can be evaluated by the semi-empirical calculation approach of Pulina (1993) which estimates the shift induced by transver sal currents. The approach is based on the balance of forces of outer hydrodynamic resistance, acceleration resistance, accelerated mass and transversal speed:

where: v_E = mean entrance velocity [m/s], L = motorcraft length [m], C_y = drag coefficient [–], T = motorcraft submergence [m], ρ= water density [kg m^{-3}], b = entrance width [m], Z = distance from entrance to fairway [m], v_S = mean motorcraft speed [m/s], m_G = total accelerated mass [kg]. It must be considered that this approach is exclusively approved for freighters.

More important than analytic calculation results are restrictions by local water authorities that state max imum inflow velocity of 0.3 ms^{-1} for the present investigation.

Waterbody 1 is a Federal Waterway Category 5 (BMVI 2015). Hence, the navigability of small freighters must not be restricted. Therefore, the local water authorities define the maximum mean velocity at the fishway's inlet as 0.3 ms^{-1} . As shown in fig ure 8 the planning state reaches the maximum velocity at v_{mean} = 0.34 ms^{-1} . The calculation of lateral move ment according to Pulina (1993) results in a movement of 0.4 m for class Johann Welker freighters travelling with a speed of 5 kmh^{-1} . Nevertheless, considering passability the planning state can not be processed due to the restrictions.

Option 1 shows a similar appearance. The mean velocity at the inlet profile is 0.39 ms^{-1} . Due to the enlarged pool, the influencing discharge volume is increased. Calculation according to Pulina (1993) yields lateral movement of more than 1 m while trav elling at a speed of 5 kmh^{-1} ,rises as traveling speed increases. As seen in figure 7 Option 2 is nearly iden tical to Option 1. This is due to the fact that the inlet

Figure 7. Velocity distribution at leading flow.

Figure 8. Comparison of velocity magnitude in different distances of the inlet pool.

pool geometry is not changed from option 1 to option 2 and shows that the flow guiding wall has little impact on the flow situation in the inlet pool.

Consequently, a flow guiding wall which separates the inflow of the fishway and the fairway leads to more safety on the waterway.

5.4 Recommendation

Considering the three evaluation criteria mentioned, option 2 represents the best solution for practical appli cation. Due to the restriction of local water authorities concerning the inflow velocity a separation of the fishway's entrance and the fairway is required to guarantee interference-free ship movement. Further investigations are necessary to evaluate the fishway's detectability. Since the fish migration corridor of the local fish population is unknown, a detailed study should be arranged to identify this corridor. Finally, option 2 can be preferred if relevant fish species migrate along the embankment.

6 CONCLUSION

Numerical simulation provides a adequate tool to eval uate hydraulic issues during the planning phase. The present investigation shows the possibility to improve the flow situation at the inlet of a fishway which cannot be a design guideline solution. However, atten tion should be paid to quality components which have a main influence on success and cost-benefit of numerical simulations. Although hydraulic investiga tions can never evaluate fishes' behavior. Therefore, it is advisable to investigate the fishes' behavior in the investigation area to evaluate the structure's final design carefully. Experiences showed that subsequent adjustments at fishways are common. Thus, it is also advisable to construct a flexible inlet structure to react

Dr. Lehners + Wittorf Consulting Engineers (L+W Cons. Eng.) (2014). Herstellung der ökologischen Durchgängigkeit zwischen Kanaltrave und Wakenitz: Entwurfsplanung. Lübeck, Germany [in German].

Oertel, M. & Schlenkhoff,A. (2012). Crossbar Block Ramps: Flow Regimes, Energy Dissipation, Friction Factors, and Drag Forces. J. Hyd. Engng., 138(5), 440–448. Pulina, B. (1993). Bestimmung der zulssigen Strömungsgröe fr seitliche Einleitungsbauwerke an Bundeswasserstraen. Mitteilungsblatt der Bundesanstalt für Wasserbau, Vol 70 [in German]. Smith, D. L.; Goodwin, R. A & Nestler, J. M. (2014). Relating Turbulence and Fish Habitat — A New Approach for Management and Research. Reviews in Fisheries Science & Aquaculture, 22(2), 123 – 130. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Characterization of flow resistance in a floodplain for varying building

density

S. Guillén-Ludeña, D. Lopez, E. Mignot & N. Rivière

Université de Lyon, INSA de Lyon, Laboratoire de Mécanique des Fluides et d'Acoustique, France

ABSTRACT: Extreme flood events with high return periods (T > 1000 years) are expected to be more common

in the next decades due climate change. In these extreme events, land occupation in floodplains becomes a major

issue as it highly influences the flow resistance and therefore, the hydrodynamic processes. The present work

aims at identifying, as the land occupation increases i) the evolution of the flood risk associated with a high

return period flood event and ii) the relative magnitude of the resistance forces acting on the flow by both the bed

friction and the drag induced by buildings. For these purposes, this paper presents an experimental methodology

to assess the influence of both the bed-friction forces and the drag forces on the overall flow resistance. Also,

the transition from a flow governed mostly by bed-friction forces to a flow in which the drag forces due to the

obstacles are predominant is studied herein.

1 INTRODUCTION

In France, the potential adverse impacts related to

flood events have increased during the last 30 years

(according to the Centre for Research on the Epidemi

ology of Disasters) because of the increasing human

settlements over the floodplains. Recent studies on

global flood risk (e.g. Hirabayashi et al., 2013) have

demonstrated that, on a global scale, the number of people exposed to floods with a return period higher than 100-year is expected to increase, irrespective of the climate models and scenarios. In this context, the assessment of the existing modeling practices for river floods with T > 100-year becomes a major issue. It is also a major issue in the context of the application of the European Flood Directive (EU, 2007) on the assessment and management of flood risks, notably to build the flood hazard maps in the areas with potential significant flood risks, and to complete the flood risk management plans.

Extreme floods are by their rare and dangerous nature characterized by a lack of field data. Flood marks are scarce, velocity measurements are non existent, and the available stage-discharge relation ships are not reliable in this range (Lang et al. 2010). Furthermore, it is usually considered that above T ~ 1000-year all the floodplains are inundated—flood protection systems are surpassed with negligible effects on the flow—so that the flow processes and the flow resistance in the floodplain are mostly controlled by land occupation.

Existing studies carried out in smooth beds (Nepf 1999; Tanino & Nepf 2008; Herbich & Shulits 1964) corroborate the influence of the spatial density of the obstacles (land occupation) on both the drag and the bed-friction forces and thus, on the overall flow resistance. In addition, Tanino & Nepf, 2008 estimated that, for smooth beds, the bed friction forces represent about 13% of the overall flow resistance, though they pointed out that in rough beds such as natural floodplains, the contribution of the bed friction to the overall flow resistance may not be negligible. This study goes beyond the aforementioned studies by presenting an experimental set-up with rough bed and with direct measurements of the drag force acting on the obstacles. The present study focuses on complex land occupation with interspersed types of hydraulic roughness and evaluates the contributions of bed-induced friction and drag due to emergent macroroughness to the global flow resistance of a floodplain. The objective is twofold: (1) to investigate the transition from a flow resistance mainly generated by the bed roughness, to a flow resistance mainly governed by the drag force created by emergent obstacles; (2) to perform a sensitivity analysis, estimating the errors made when neglecting the presence of the obstacles, considering only the bed roughness. This paper presents the experimental methodology followed in order to reach the objectives mentioned above. 2 EXPERIMENTAL SET-UP 2.1 Experimental facility The experiments are carried out in a 1.20 m width and 8 m long rectangular laboratory flume with a fixed slope of $S_0 = 0.18\%$. The bottom of the flume

Figure 1. Upstream view of the experimental flume.

is covered by an artificial grass layer, onto which the

emergent obstacles are placed (Fig. 1).

The obstacles consist of prismatic bricks with

dimensions: 0.054×0.054 m ^{2} along both horizon

tal directions and remain emergent in all cases. The

upstream end of the flume is equipped with a hon

eycomb panel to tranquilize the flow (Fig. 1). At the

downstream end of the flume the flow depth is con

trolled by means of an adjustable tailgate. The flume

is equipped with a robotic arm that allows displace ments in the three directions: X , Y , and Z (Fig. 1).

This robotic arm allows us to measure water depth by means of an ultrasonic limnimeter and velocities by using a micro-ADV.

2.2 Experimental parameters

Dimensional analysis yields the following parameters as characteristics of the case of study:

where B is the channel width (B = 1.20 m), L stands for the distance between the centers of two consecutive obstacles in both directions (X and Y) (Fig. 2) and the ratio B/L thus denotes the number of bricks per cross-section. The spatial distribution of the bricks is parameterized by the spatial density λ defined as:

where d is the flow depth and w is the characteris tic dimension of the bricks (w = 0.054 m) (Fig. 2). Re is the Reynolds number of the flow considering d as length scale, ε is the bed roughness (ε= 0.007 m), the ratio ε/d denotes the relative roughness, and $Fr = U/(g \cdot d)^{1/2}$ is the Froude number, where g is the gravitational acceleration.

In this study, only fully turbulent flows with low Froude number ($Fr < 0.30$) are considered.These con siderations allow us to neglect the effects of Re and Fr,

Figure 2. Schematic plan view of the obstacle distribution.

Table 1.

B/L	λ	d/L	ε/d
5 to 11	0.15 to 0.25	0.2 to 1.1	0.03 to 0.17

which reduces to 4 the number of non-dimensional parameters, namely

2.3 Planned range of parameters

In this study we will consider values of the filling ratio B/L ranging from 5 to 11 (typically 8 cases). For each value of B/L, three values of lambda and three values of d/L will be studied. Table 1 contains the typical ranges of the different non-dimensional parameters planned in the experiments. 2.4 Experimental procedure and measurements Once the obstacles are placed throughout the flume, the corresponding flow depth (d) is set by means of the downstream tailgate and by adjusting the flow discharge until reaching uniform-flow condition. This condition is verified by means of topographic surveys of the water surface all along the flume. When the uniform flow is set, the drag force (F_d) is measured by means of a hydrodynamic balance, which is schematized in Figure 3. In this balance the drag force (F_d) is multiplied by a lever arm whose top end is attached to a weir (Fig. 3). This weir pulls a calibrated weight that is on a digital scale (Fig. 3), which measures the multiplied force (Fig. 3). To determine the point where the drag force acts on the obstacle, each measurement is performed twice by changing the

Figure 3. Schematic view of the hydrodynamic balance.

axis of rotation of the lever arm. The drag force (F_d) is obtained by applying a moment balance around the two axis separately.

Also under uniform flow conditions, water surface and 3D velocity field are characterized in detail for one cell located far enough from boundary conditions (Fig. 2). An ultrasonic limnimeter with an accuracy of ±1 mm is used to measure the water surface and a side-looking Nortek-Vectrino (micro-ADV) is used to measure the 3D velocity field.

3 RESULTS

3.1 Methodology

The strategy consists in measuring the drag force (F_d) that the flow exerts on one obstacle located at the middle of a cell (see Fig. 2). Once the value of F_d is known, the bed-friction force (F_b) can be obtained by applying momentum balance to the measured cell, which performed along the streamwise direction and under uniform flow conditions, results in the following expression:

where F_d is the drag force, F_b is the bed-friction force, and W stands for the weight component of the water. Equation 4 indicates that, under uniform flow conditions, the resultant force acting on the fluid ($F_d + F_b$) is compensated by the weight component of the fluid (W).

Since F_d is measured and the weight component of the fluid (W) is obtained as

where ρ is the density of the water, and g is the grav itational acceleration, the bed-friction force (F_b) can

Herbich, JB, and Shulits, S. 1964. “Large Scale Roughness in Open-Channel Flow.” Proceedings of the American Soci ety of Civil Engineers. Journal of the Hydraulics Division, 203–30.

Hirabayashi, Y, Mahendran, R, Koirala, S, Konoshima, L, Yamazaki, D, Watanabe, S, Kim, H, and Kanae, S. 2013. “Global Flood Risk under Climate Change.” Nature Cli mate Change 3 (9): 816–21. doi:10.1038/nclimate1911.

Lang, M, Pobanz, K, Renard, B, Renouf, E, and Sauquet,

E. 2010. “Extrapolation of Rating Curves by Hydraulic Modelling, with Application to Flood Frequency Analysis.” Hydrological Sciences Journal 55 (6): 883–98. doi:10.1080/02626667.2010.504186. Nepf, HM. 1999. “Drag, Turbulence, and Diffusion in Flow through EmergentVegetation.”Water Resources Research 35 (2): 479–89. doi:10.1029/1998WR900069. Tanino,Y, and Nepf, HM. 2008. “Laboratory Investigation of Mean Drag in a Random Array of Rigid , Emergent Cylinders.” Journal of Hydraulic Engineering 134 (January): 34–41. doi:10.1061/(ASCE)HY.1943-7900.0000080. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Turbulent dispersion in bounded horizontal jets. RANS capabilities and physical modeling comparison

D. Valero

FH Aachen University of Applied Sciences, Aachen, Germany

University of Liege (ULg), Liège, Belgium

D.B. Bung

FH Aachen University of Applied Sciences, Aachen, Germany

M. Oertel

FH Lübeck University of Applied Sciences, Lübeck, Germany

ABSTRACT: Accurate prediction of contaminant concentrations can yield improved outfall structures design, reducing economical expenses and environmental impact. Conventionally, cost intensive physical modeling or simplified integral approach models have been employed despite their drawbacks and limitations. In the present paper, two jet setups have been studied by means of 3D Reynolds Averaged Navier-Stokes equations and experimental modeling. Both jet cases correspond to turbulent horizontal jets, bounded by the channel bed,

which might be found in common environmental discharges. Three of the most widely employed turbulence models up to date have been investigated (namely standard k-ε, RNG k-ε and k-ω), analyzing their performance on the jet trajectory estimation. For the best performance's model, RNG k-ε and for both jets cases of the present study, analysis has been extended to the turbulence diffusion estimation by defining a turbulent Schmidt number.

1 INTRODUCTION

Outfall structures are present worldwide, affecting local and global overall environmental conditions. Proper design of environmental flow discharges is of paramount importance for the environmental systems' biotic media. Improving knowledge on numerical modeling of such structures may reduce costs yielding altogether safer designs.

Traditionally, outfalls have been designed by means of physical modeling. However, due to potential scale effects (i.e.: when dealing with different densities between water body and jet) in small scale models and the large space requirements necessary for a river reach model (since vertical model distortion should be avoided to properly reproduce buoyant jets; Kobus 1980), numerical methods may arise as a feasible and yet reliable tool.

When using a numerical model, practitioners are usually attracted to the integral method approach, which is subject to considerable restrictions due to model's hypothesis. However, they hold an extensive experience and validation (Jirka 2004, Jirka 2007, Bleninger & Jirka 2008, Loya-Fernández et al. 2012, Palomar et al. 2012). Main limitations of these models (as restriction to unbounded jets and no contaminant re-entrainment among others) can be found in Jirka (2004, 2007). With increasing computer capacities, more complex models have arisen. 20 years ago it was not possible to simulate complex river reaches – nowadays, new hardware advances makes it feasible at affordable costs. Thus, Reynolds Averaged Navier-Stokes (RANS) equations appear to be the more common approach in Computational Fluid Dynamics (CFD) when a 3D approach is desired. Large Eddy Simulations (LES) can be already performed and may yield to more accurate and detailed results (Ruiz et al. 2015). However, modeling large domains of interest such as in environmental discharges problems or complex multi-physics (Gorlé & Iaccarino 2013) might be restricted to RANS modeling. Additionally, as mentioned by Piomelli (2014) the advanced level of competence required to run a LES is an obstacle to its widespread application. In this study, three of the most commonly used turbulence models have been used in order to obtain jet trajectories for two horizontal discharge cases. Furthermore, the turbulence model with best performance has been used in order to analyze turbulent dispersion of both two jets. The studied flow cases represent two bed bounded discharges which aim to be representative of a wide range of outfall structures.

2 EXPERIMENTAL SETUP

Experimental model runs were conducted at FH Aachen's Hydraulic Laboratory. A horizontal flume of 0.58 m width has been used for physical model tests (Figure 1). Inside, two diffusors have been adopted, allowing horizontal discharges at 90 and 45 degrees with the freestream flow direction (ϕ 0), as shown in

Figure 2. Inner diameter of the injector is 3 mm, which

is connected to a pressurized tank allowing a constant

pressure head and thereby a constant flow discharge.

The diffusor is located at 8 mm from the channel bed

and 80 mm from the channel wall.

Figure 1. Physical model setup, pressurized tank above

chute and inlet boundary condition at the left side of the

image.

Figure 2. Sectional side view of the experimental setup

corresponding to the simulated cases, ambient flow from

left to right and markers every 5 cm, top: $\phi_0 = 90^\circ$, bottom:

$\phi_0 = 45^\circ$. Average flow velocity corresponds to 2 m/s, which allows an efflux Reynolds number in the order of 6,000; being over 2,000 commonly accepted as the critical value for the jet to be independent of viscosity in a turbulent ambient flow (Fischer et al. 1979, Kobus 1980). Ambient water flow rate was set to 13 l/s by use of a frequency regulator and an electromagnetic flowmeter. Flow depth was controlled with a downstream weir and set to 0.28 m at the flow region of interest, yielding to an average ambient velocity of 0.08 m/s. The role of ambient flow is to gradually deflect the turbulent jet into the flow direction while inducing additional mixing. The effluent water has been dyed by adding potassium permanganate (dilution: 1 g per 9 l water) to allow visual inspection of the jet trajectory, envelopes boundaries and mixing. A Canon EOS 7D camera was used to capture the investigation domain from side and top views. 3 NUMERICAL MODELING 3.1 General settings FLOW-3D has been used to set up and run numerical simulations. RANS equations are discretized by means of the Finite Volume Method (Versteeg & Malalasekera 2007), resulting in a linear system which is numerically solved by using the Krylov projection based method GMRES (Saad 2003). For advection flux approximation, a second order explicit scheme was employed including a slope limiter algorithm ensuring monotonicity preserving of the advected quantities (Van Leer 1977, Hirsch 2007). Diffusion is computed explicitly and a CFL condition is set to 0.75 for dynamically adjusting time step limit without compromising numerical stability. Turbulent viscosity was calculated by using three of the most widely used turbulence models. The selected turbulence models are all two equations based models, which are the first complete models able to reproduce all sorts of turbulence (Pope 2000). However, no turbulence model can reproduce properly all types of turbulent flows and hereby validation becomes necessary. First employed turbulence model corresponds to the standard k-ε (Pope 2000, Davidson 2015) which uses a transport equation for the turbulent kinetic energy (k) and one for its dissipation (ε). The second model is the RNG k-ε, as defined by Yakhot & Orszag (1986) and Yakhot et al. (1992), which is based on the Renormalization Group theory (McComb 2004) and adds an additional term to the ε equation and uses different coefficients for the parameters.And the third one is the k-ω turbulence model, as defined by Wilcox (1998, 2008), which substitutes the equation for ε by another for ω= εk −1 . This model is known to perform superiorly for many types of flows when compared to the standard k-ε (Pope 2000) although it has not been as

widely tested as k-ε model (Bradshaw et al. 1996).

Furthermore, it has been previously noticed that k-ε significantly overpredicts spreading of the round jet (Pope 2000). Otherwise, k-ε model is known to be more insensitive to free stream values. In the question of the choice of a second variable in two equations models, freestream sensitivity should be given high priority (Spalart 2000). Selection of RNG k-ε is based on the experience of previous researchers which noted that RNG k-ε reproduces flow separation better than the standard k-ε (Speziale & Thangam 1992, Brad shaw 1997, Kim & Baik 2004, Bung et al. 2008) and has shown better performance also for impinging jets (Dutta et al. 2013, Valero & Bung 2016).

For an in-depth discussion on turbulence models performance, strengths and limitations, reader might be addressed to Pope (2000), Spalart (2000), Wilcox (1998) and Bradshaw et al. (1996).

3.2 Turbulent diffusion

Transport and diffusion of a scalar C can be modeled by using:

where V_F is the cell volume fraction and A_i is the cell area in i-direction. Accordingly, u_i is the velocity component in idirection. This formulation is already accounting for FAVOR method, which is employed by FLOW-3D for the definition of obstacles in the meshed domain (Hirt & Sicilian 1985). D accounts for the total diffusion (both molecular and turbulent). But as far as turbulent mixing happens, it is reason able to assume that turbulence transport will prevail over molecular diffusion and thus $D \approx D_t$ can be con sidered. Definition of this turbulent diffusion requires an additional relation. Herein, the so-called gradient diffusion hypothesis is assumed (Pope 2000). Gradi ent diffusion hypothesis estimates that the advective transport of a scalar due to turbulent fluctuations is related to the spatial gradient of the concentration by means of a turbulent diffusivity.Then, definition of the turbulent Schmidt number (Sc_t) becomes necessary:

where ν_t is the turbulent viscosity. Use of a two

equation turbulence model allows estimation of this

turbulent viscosity as follows:

For both k-ε based models, being $C_\mu = 0.090$ for

the standard version and 0.085 for the RNG. In the

case of k-ω, it can be calculated similarly: This turbulent viscosity, additionally to its relation to the turbulent diffusion, plays a key role on the estimation of the mean velocity field, and thus the jet trajectory. As noted by previous authors, Sc_t appears to be of major importance for modeling turbulent transport in RANS models (Spalding 1971, He et al. 1999, Tominaga & Stathopoulos 2007, Gualtieri & Bombardelli 2013). Performance assessment of more complex scalar transport models can be found in Tominaga & Stathopoulos (2013) and Rossi & Iaccarino (2009). The early CFD study of Spalding (1971) recommended the Sc_t value of 0.7 for a round turbulent free jet, which has been later employed in different numerical studies (Li & Stathopoulos 1997, Wang & McNamara 2006). However, employed values usually range from 0.2 to 1.5 (Tominaga & Stathopoulos 2007, Gualtieri & Bombardelli 2013, Valero & Bung 2016). Additionally, Koeltzsch (2000) analyzed classic experimental studies on boundary layer flows noticing that turbulent Schmidt number values range from 0.5 to 0.9 although local value depends on height over the wall. It is often reported higher values for numerical modeling than the experimentally based ones. 3.3 Meshing and boundary conditions Multimesh approach has been used in order to selectively refine the cell size close to the Zone of Flow Establishment (ZOFE), as shown in Figure 3. Ordered from coarser to finer, meshes are described in Tables 1 and 2. Meshes for case $\phi_0 = 45^\circ$ are similar to case $\phi_0 = 90^\circ$ but with extended x dimension to better capture the obliquus injector flow. Meshes 1and 2 cover the complete domain, as shown in Figure 3. The extension of the mesh 1 is 2.0 m, ensuring self-development of the ambient flow and thus auto adjustment of turbulent quantities in the flow approaching the diffuser. Length of mesh 2 is 0.60 m, which covers the region studied in the physical model. Width of both meshes corresponds to 0.58 m and height to 0.28 m, which are the width and depth of the chute flow. Meshes 1 and 2, which are linked, cover the 28 cm of flow depth with finer cells closer to the channel bed. Meshes 4 to 6 are extended in the vertical direction only to cover the whole jet boundaries. Figure 3. Sectional side view of meshing and main boundary conditions for case $\phi_0 = 90^\circ$.

Table 1. Mesh description for case $\phi_0 = 90^\circ$.

Mesh	Minimum dimensions [mm]			Number of cells
	Δx	Δy	Δz	
#1*	20	20	20	40,600
#2	10	10	2	129,920
#3	5	5	2	432,000
#4	3	3	2	116,400
#5	0.64	1.5	1.5	68,900
#6**	0.64	1	1	1,520

* Buffer mesh, for self-development of ambient flow.

** Injector mesh, for accurate discretization of inner injector pipe.

Table 2. Mesh description for case $\phi_0 = 45^\circ$.

Mesh	Minimum dimensions [mm]			Number of cells
	Δx	Δy	Δz	
#1*	20	20	20	40,600
#2	10	10	2	129,920
#3	5	5	2	432,000
#4	3	3	2	228,780
#5	0.64	1.5	1.5	213,720
#6**	0.64	1	1	35,020

* Buffer mesh, for self-development of ambient flow.

** Injector mesh, for accurate discretization of inner injector pipe.

At channel bed and laterals, smooth wall boundary conditions are considered. For the flow inlet at mesh 1, a constant velocity is set to the value of the ambient water flow (u_a in Fig. 3). Downstream the boundary condition, placed at mesh 2, hydrostatic pressure is imposed (p_a in Fig. 3). For free surface, a symmetry boundary condition is stablished. At mesh 6, mean injector velocity is specified altogether with a scalar concentration C = 1.0.

The above described mesh setup is the result of an iterative process where cells were refined until no noticeable change occurred in the jets' trajectories and boundaries.

4 RESULTS AND DISCUSSION

4.1 General remarks

Shown boundaries in the numerical model correspond to C = 0.01 isosurfaces, which may correspond roughly to the visual observations as described in Valero & Bung (2016) for a similar setup.

In Section 4.2 jet trajectories are shown for all three tested turbulence model (k-ε, RNG k-ε and k-ω) without any turbulent dispersion ($D_t = 0.0$ or $Sc_t = \infty$). In Section 4.3 jet boundaries for different Sc_t numbers are shown for the RNG k-ε turbulence model. Figure 4. Case $\phi_0 = 90^\circ$ solution with no turbulent dispersion. Figure 5. Case $\phi_0 = 45^\circ$ solution with no turbulent dispersion. 4.2

Jet trajectories Jet trajectories are dependent on the proper computation of turbulent viscosity, which depends on the turbulence model. Solution has shown to be very sensitive to refinement in the ZOFE.Therefore, meshes have been added and refined covering this entire region until no significant change in the solution was noticed. Meshes have been located around the jet boundaries and have been extended up to the point were no change was produced over the jet trajectory. Case $\phi_0 = 90^\circ$ (shown in Fig. 4) has proved to be more challenging for the turbulence models, which might be explained by the larger curvature of the streamlines resulting from the crossflow effect. It is shown that all three turbulence models overestimate the reach of the jet, being the standard k-ε the one differing more from the physical model. Otherwise, RNG k-ε is showing better agreement than the others. For case $\phi_0 = 45^\circ$ (shown in Fig. 5), both standard and RNG k-ε turbulence models yield to similar results. A larger difference can be observed for

Figure 6. Case $\phi_0 = 90^\circ$ solution for RNG k-ε and different turbulent Schmidt numbers.

k-ω, where even not considering turbulent diffusion is resulting in wider jet boundaries.

4.3 Jet boundaries

As discussed in Section 3.2, relative position of jet boundaries to the centerline trajectory depends upon the turbulent diffusion, which is controlled by turbu lent Schmidt number Sc_t. In the present study, values from 0.7 to 1.6 have been investigated for both hori zontal jets altogether with RNG k-ε turbulence model. For sake of clarity, only solutions related to values 0.7 and 1.4 are shown. First value represents the original recommendation of Spalding (1971) and the physically based values recommended byYimer et al. (2002); and

1.4 is the best fit obtained by Valero & Bung (2016) for jets in a crossflow.

For case $\phi_0 = 90^\circ$, major differences arise from the computation of the centerline as discussed in Section 4.2, which is directly related to the turbulence model. As shown in Figure 4, differences increase for the remaining tested turbulence models. However, width of the jet is very similar for both cases in the far field region. Differences are noticeable only in the region with higher curvature of the jet.

For case $\phi_0 = 45^\circ$, where jet trajectory was already well reproduced by the RNG k-ε turbulence model, boundaries fit better to the experimental results. Since it is not an appreciable difference between $Sc_t = 0.7$ and $Sc_t = 1.4$, it is clear that using a value $D_t \neq 0$ improves results quality, as shown in Tables 3 to 6.

For both cases, despite trajectory differences in case $\phi_0 = 90^\circ$, results are in good agreement with the physical model. Width of the contaminant spread has shown a relative absolute maximum error of 10.16% in the far field (x = 0.50 m) for $Sc_t = 0.7$, 7.08% for $Sc_t = 1.4$ and 34.07% for $D_t = 0.0$; thus showing importance of accounting for turbulence dispersion in the RANS modeling of an environmental discharge. More remarkable differences occur in the near field where turbulence quantities are larger. Figure 7. Case $\phi_0 = 45^\circ$ solution for RNG k-ε and different turbulent Schmidt numbers. Table 3. Relative errors [%] of the jet boundary width at x = 0.50 m. Case $Sc_t = 0.7$ $Sc_t = 1.4$ $D_t = 0.0$ $\phi_0 = 90^\circ$ -4.63 -0.45 -9.38 $\phi_0 = 45^\circ$ 10.16 7.08 -34.06 Table 4. Relative errors [%] of the jet boundary width at x = 0.40 m. Case $Sc_t = 0.7$ $Sc_t = 1.4$ $D_t = 0.0$ $\phi_0 = 90^\circ$ -6.87 -5.44 -12.40 $\phi_0 = 45^\circ$ 3.27 1.96 -39.78 Table 5. Relative errors [%] of the jet boundary width at x = 0.30 m. Case $Sc_t = 0.7$ $Sc_t = 1.4$ $D_t = 0.0$ $\phi_0 = 90^\circ$ 1.98 3.90 -10.20 $\phi_0 = 45^\circ$ -4.00 -13.46 -52.25 Table 6. Relative errors [%] of the jet boundary width at x = 0.30 m. Case $Sc_t = 0.7$ $Sc_t = 1.4$ $D_t = 0.0$ $\phi_0 = 90^\circ$ 14.05 9.78 -16.55 $\phi_0 = 45^\circ$ -3.32 -18.42 -60.14 5 CONCLUSIONS In this study, RANS capabilities for environmental discharges modeling have been analyzed. Three different

turbulence models have been tested and compared with

physical model results of two horizontal jet bounded

flow cases. It has been shown that RNG k-ε is able to

better reproduce jet trajectories despite some differ

ences while definition of a turbulent Schmidt number

becomes necessary in order to account for turbulence

dispersion. Usage of a turbulent Schmidt number

when properly selected – reduces relative errors in the

jet boundary width one order of magnitude, generally

yielding to errors less than 10% for such variable.

It is remarkable that turbulence model selection

induces a major change in overall result and thus

special care is necessary. Additionally, when using

k-ω, which has shown larger unsteadiness, different

turbulent Schmidt numbers might be expected. This

unsteadiness can surpass the effect of the real turbulent

dispersion precluding use of any turbulent Schmidt value.

Consequently, use of different turbulent Schmidt numbers may help assess uncertainty of RANS numer ical modeling in the design of complex outfall struc tures. Capabilities of the employed models are wider than integral model approach with a reasonable com putational cost (when compared to more cost intensive techniques, i.e.: LES).

Bleninger, T. & Jirka, G.H. 2008. Modelling and environ mentally sound management of brine discharges from desalination plants. Desalination, 221(1), 585–597.

Bradshaw, P., Launder, B.E., & Lumley, J.L. 1996. Collab orative testing of turbulence models. Journal of Fluids Engineering, 118(2), 243–247.

Bradshaw, P., 1997. Understanding and prediction of turbu lent flow—1996. International Journal of Heat and Fluid Flow, 18(1), 45–54, ISSN: 0142-727X.

Bung, D.B., Hildebrandt, A., Oertel, M., Schlenkhoff, A. & Schlurmann, T. 2008. Bore propagation over a submerged horizontal plate by physical and numerical simulation. Proc. 31 st Intl. Conf. Coastal Eng., Hamburg, Germany, 3542–3553.

Davidson, P. 2015. Turbulence: an introduction for scientists and engineers. Oxford University Press, USA.

Dutta, R., Dewan, A. & Srinivasan, B. 2013. Comparison

of various integration to wall (ITW) RANS models for

predicting turbulent slot jet impingement heat transfer.

International Journal of Heat and Mass Transfer, 65,

750–764.

Fischer, H.B., List, E.J., Koh, R.C.Y., Imberger, J. & Brooks,

N.H. 1979. Mixing in Inland and Coastal Waters. Aca

demic Press, New York, NY.

Gorlé, C., & Iaccarino, G. 2013. A framework for epistemic

uncertainty quantification of turbulent scalar flux mod

els for Reynolds-averaged Navier-Stokes simulations.

Physics of Fluids 25 055105. http://dx.doi.org/10.1063/

1.4807067

Gualtieri, C., & Bombardelli, F. 2013. Parameterization of the

turbulent Schmidt number in the numerical simulation of

open-channel flows. XXXV IAHR Congress.

He, G., Guo, Y., & Hsu, A. T. 1999. The effect of Schmidt

number on turbulent scalar mixing in a jet-in-crossflow.

International Journal of Heat and Mass Transfer, 42(20),

3727–3738. Hirsch, C. 2007. Numerical Computation of Internal and External Flows: The Fundamentals of Computational Fluid Dynamics: The Fundamentals of Computational Fluid Dynamics. Butterworth-Heinemann. Hirt, C.W., & Sicilian, J.M. 1985. A porosity technique for the definition of obstacles in rectangular cell meshes. In Proc. Fourth International Conf. Ship Hydro (pp. 1–19). Jirka, G.H. 2004. Integral model for turbulent buoyant jets in unbounded stratified flows. Part I: Single round jet. Environmental Fluid Mechanics, 4(1), 1–56. Jirka, G.H.

2007. Buoyant surface discharges into water bodies. II: jet integral model. Journal of Hydraulic Engineering, 133(9), 1021–1036. Kim, J.J., & Baik, J.J. 2004. A numerical study of the effects of ambient wind direction on flow and dispersion in urban street canyons using the RNG k-ε turbulence model. Atmospheric Environment, 38(19), 3039–3048. Kobus, H. 1980. Hydraulic modelling. Bulletin 7, German Association for Water Resources and Land Improvement. Parey, Hamburg. Koeltzsch, K. 2000. The height dependence of the turbulent Schmidt number within the boundary layer. Atmospheric Environment, 34(7), 1147–1151. Loya-Fernández, Á., Ferrero-Vicente, L.M., Marco-Méndez, C., Martínez-García, E., Zubcoff, J. & Sánchez-Lizaso, J.L. 2012. Comparing four mixing zone models with brine discharge measurements from a reverse osmosis desalination plant in Spain. Desalination, 286, 217–224. McComb, W.D. 2004. Renormalization Methods. A Guide for Beginners. Oxford University Press. Palomar, P., Lara, J.L., & Losada, I.J. 2012. Near field brine discharge modeling part 2:Validation of commercial tools. Desalination, 290, 28–42. Piomelli, U. 2014. Large eddy simulations in 2030 and beyond. Philosophical Transactions of The Royal Society A, 372: 20130320. http://dx.doi.org/10.1098/rsta. 2013.0320 Pope, S.B. 2000. Turbulent Flows. Cambridge University Press. Rossi, R. & Iaccarino, G. 2009. Numerical simulation of scalar dispersion downstream of a square obstacle using gradient-transport type models. Atmospheric Environment, 43(16), 2518–2531. Ruiz, A.M., Lacaze, G. & Oefelein, J.C. 2015. Flow topologies and turbulence scales in a jet-in-crossflow. Physics of Fluids, 27, 045101 2015. DOI: http://dx.doi.org/10.1063/1.4915065 Saad, Y. 2003. Iterative methods for sparse linear systems. Siam. Spalart, P.R. 2000. Strategies for turbulence modelling and simulations. International Journal of Heat and Fluid Flow, 21(3): 252–263. Spalding, D.B. 1971. Concentration fluctuations in a round turbulent free jet. Chemical Engineering Science, 26(1), 95–107. Speziale, C.G. & Thangam, S. 1992. Analysis of an RNG based turbulence model for separated flows. International Journal of Engineering Science, 30(10), 1379-IN4. Tominaga, Y. & Stathopoulos, T. 2007. Turbulent Schmidt numbers for CFD analysis with various types of flow field. Atmospheric Environment, 41(37), 8091–8099. Tominaga, Y. & Stathopoulos, T. 2013. CFD simulation of near-field pollutant dispersion in the urban environment: A review of current modeling techniques. Atmospheric Environment, 79, 716–730. Van Leer, B. 1977. Towards the ultimate conservative difference scheme. IV.A new approach to numerical convection. Journal of computational physics, 23(3), 276–299.

Valero, D. & Bung, D.B. 2016. Sensitivity of turbulent

Schmidt number and turbulence model to simulation of

jets in crossflow. Environmental Modelling & Software

(accepted). DOI: 10.1016/j.envsoft.2016.04.030

Versteeg, H.K., & Malalasekera, W. 2007. An introduction to

computational fluid dynamics: the finite volume method.

Pearson Education.

Wilcox, D.C. 1998. Turbulence modeling for CFD. La

Canada, CA. DCW industries.

Wilcox, D.C., 2008. Formulation of the k-omega turbulence

model revisited. AIAA Journal (46), 2823–2838.

Yakhot,V. & Orszag, S.A. 1986. Renormalization group anal

ysis of turbulence. I. Basic theory. Journal of scientific

computing, 1(1), 3–51. Yakhot, V., Orszag, S.A., Thangam, S., Gatski, T.B. & Speziale, C.G. 1992. Development of turbulence models for shear flows by a double expansion technique. Physics of Fluids A: Fluid Dynamics (1989–1993), 4(7), 1510–1520. Yimer, I., Campbell, I. & Jiang, L.Y. 2002. Estimation of the turbulent Schmidt number from experimental profiles of axial velocity and concentration for high-Reynoldsnumber jet flows. Canadian Aeronautics and Space Journal, 48(3), 195–200. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Modelling methane emissions from Vilar reservoir (Portugal)

R.F. Lino, P.A. Diogo, A.C. Rodrigues & P.S. Coelho

MARE, FCT-UNL, Portugal

ABSTRACT: The decomposition of the sediments in the benthic zone of reservoirs may induce the production

of greenhouse gases which end up being released to the atmosphere. This study focused on the quantification

of methane emissions to the atmosphere from Vilar reservoir using the model CE-Qual-W2 as methane emis

sions importance is strongly associated with its global warming potential, 25 times higher than carbon dioxide.

According to Beaulieu et al. (2014), methane emissions from reservoirs in tempered climates are usually below

50 mg $C.m^{-2} d^{-1}$ and in tropical climates between 50 and 150 mg $C.m^{-2} d^{-1}$. Considering the reliability and

experience in the use of the CE-Qual-W2 model, it is considered that although field data limitations are yet a

problem to be tackled, the implementation of this model is surely a way to go towards a better understanding of

the potential contribution from reservoirs to GHG emissions.

1 INTRODUCTION

1.1 Methane emissions in worldwide reservoirs

Hydroelectricity has been highly implanted worldwide due to its high energy production capacity but although it is a renewable energy source, the construction of hydropower dams is far from being a clean solution: landscape destruction, flooding and altering the natu ral course of rivers caused by dams, as well as water quality problems are some of the issues related to dam construction. Furthermore, the decomposition of the sediments in the benthic zone of reservoirs may induce the production of greenhouse gases which end up being released to the atmosphere.

Methane emissions to the atmosphere represent a relevant contribution to global warming. Being consid ered as one of the most dangerous greenhouse gases, methane has a global warming potential twenty-five times of the CO_2 on a 100 year timescale (Forster et al., 2007).

Nowadays reservoirs occupy an area of about 5×10^5 km^2, which consists in a third of the total dimension of all natural lakes (Wildi, 2010). One of the reasons behind the growth of these reservoirs lies in the construction of hydroelectric dams due to its high energy production capacity resorting to a renew able energy source. However, uncertainties regarding greenhouse gas emissions from reservoirs raise ques tions about its contribution to global warming (Gunkel, 2009).

Methane emissions in reservoirs are highly depen dent on the trophic level of the waterbody. The degra dation of the organic matter present in the bottom of the reservoir under anaerobic conditions is the core of methane production (Adams, 2005). Residence time is also an important factor in methane production, since it influences the dissolved oxygen concentration – the longer the residence time, the less dissolved oxygen in the system (Fearnside, 2005). The water retained by dams flood vast areas covered in vegetal species, drowning them, losing their CO_2 assimilation capacity and accumulating their organic matter in the bottom of the new reservoir. This initial organic matter adds up with the sediments transported by the rivers, accumulating large amounts of organic matter that, once start degrading, end up releasing methane under anaerobic conditions and carbon dioxide in aerobic condition. Once the water gets saturated, bubbles are created to transport the gases directly to the atmosphere. Also, the organic carbon present in the flooded soils depends on its type, for example, peatlands create bigger carbon reserves, granting them a higher emission potential than other types of soli (Louis et al., 2000). However, Huttunen et al. (2002) compared two reservoirs over two different soil types and concluded that there was no direct correlation between the methane emissions from the reservoirs and the type of soil, giving bigger emphasis to the ecosystem productivity. The concentration of dissolved methane in reservoirs also depends on its stratification, meaning that the same waterbody can reach high concentrations during hotter seasons and have practically zero methane during colder/dry seasons (Abril et al., 2005). Yang et al. (2014) studied the emissions in several reservoirs worldwide and noted that reservoirs in tropical climates have the higher emissions than the ones in temperate climates (Table 1). According to Beaulieu et al. (2014), methane emissions from reservoirs in tempered climates are usually below 50 mg $C.m^{-2} d^{-1}$ and in tropical climates between 50 and 150 mg $C.m^{-2} d^{-1}$, confirming the

Table 1. CH_4 emissions from tropical and temperate reservoirs.

Climate	Country	Reservoir	Emissions mg $C.m^{-2} d^{-1}$
Tropical	French Guiana	Petit Saut	11,2–800
	Panama	Gatun Lake	526,3
	Brazil	Miranda	164,5
Temperate	Laos	Nam Theun 2	40
	Canada	Laforge	0.14
	Poland	Wilcza Wola	32–451
	Finland	Porttipahta	3.5
	Switzerland	Lake Wohlen	15

Figure 1. Vilar reservoir's drainage basin.

Table 2. Vilar reservoir characteristics.

Max pool level (m)	555
Normal pool level (m)	552
Min pool level (m)	525
Total storage (hm^3)	99,75
Perimeter (km)	148,2
Length (km)	10,4

Surface area – NPL (km^2) 6,7

Crest length (m) 240

premise of the climate being an important factor regarding methane emissions from reservoirs.

1.2 Study site

Vilar reservoir is a reservoir that resulted from the construction of a hydroelectric dam near the village of Vilar, in the north of Portugal. The reservoir is located in the Távora river, which is an affluent of Douro river and its drainage basin (Figure 1) covers a land extension of 360 km^2 in which most of the soil is either agricultural or agroforestry. The geographic location of the reservoir in the northern interior of Portugal is associated with temperate climates, having an annual average temperature of 11,1 ∘ C, varying between −2,3 ∘ C and 25,6 ∘ C. The annual mean rainfall depth of 1030 mm varies between 400 mm and 2500 mm and there’s an average of 250 rainy days per year. It is considered a high nebulosity zone due to the average 271 cloudy days per year and the wind is generally weak, predominating northwest winds.

2 METHODS AND DATA

2.1 Water quality model

Hydrodynamic and water quality simulations were carried out using CE-QUAL-W2 model, version 4.0 alpha. This model is a widely used two dimensional, laterally averaged hydrodynamic and water quality model developed at the U.S.Army Corps of Engineers, considered best suited for relatively long and narrow water bodies exhibiting longitudinal and vertical gradients (Cole & Wells, 2015). Model structure allows multiple reservoir branches and multiple different structures for withdrawals and outlets, making possible the implementation to the studied reservoir. Water quality formulations in CEQUAL-W2 allow modelling different water quality parameters like temperature, nutrients, dissolved oxygen, organic matter and sediment relationships. In terms of hydrodynamic processes and calculations, this model can formulate water surface elevations, storage volumes, vertical and horizontal velocities and temperature. Model implementation requires input data regarding inflows and outflows, water quality and meteorology. The implementation of CE-QUAL-W2 is based on a mathematical bathymetric representation composed of layers (vertical axes) and segments (longitudinal axes). For Vilar reservoir were assumed three branches composed of 500 m long segments and one meter high layers. Segment width was defined according to the available topography. The result was a grid with 51 layers and 31 segments, with the main branch having 20 segments, the second having 6 and the third having 4 segments. 2.2 Meteorological data The model CE-QUAL-W2 requires the input of meteorological data regarding air temperatures, due point, cloud cover and wind speed and direction. All these parameters were provided as monthly averages calculated from the data from SNIRH (http://snirh.apambiente.pt/, 1/7/2015). All the collected data consisted in daily measured parameters from the meteorological station of Junqueira (located 50 km northeast from the reservoir), which was the closest station with enough data to cover all the required parameters.

Figure 2. Inflow and outflow to theVilar-Tabuaço reservoir.

2.3 Flow data

The hydrological regime of the reservoir was char acterized based on daily inflows, outflows, water levels and stored volumes. All these parameters were obtained from the hydrometric station of Vilar Tabuaço and water evaporation volumes were assumed included in the water balance. Inflows and out flows are presented in Figure 2, showing evidence of hydropower dam type operation.

2.4 Water quality data

Inflow water quality data is not available for the upstream inflows to the simulated reservoir. As alter native, water quality data from a downstream monitor ing station was used to characterize reservoir inflows

(Moinho da Ponte Nova).

Inflow water quality was characterized on the basis of monthly average data for temperature, dissolved oxygen, total dissolved solids, inorganic suspended solids, PO_4, SO_4, NH_4, $NO_3 + NO_2$, CBOD, Organic matter, total inorganic carbon, alkalinity.

2.5 Simulation scenario

The simulation scenario corresponds to the actual calibration period of the model because the original objective is the quantification of the methane emis sions of the reservoir and not variations under different scenarios. Choosing the scenario was a process based on the identification of hydrological dry, wet and average years.

Data regarding precipitation was gathered from the weather station of Leomil and results indicated there were no wet years matching the period of available flow data, thus only dry or average hydrological years could be analyzed.

Eventually the years 2002/2003 (dry) and 2003/2004 (average) were chosen, being the most recent years meeting all modelling data requirements. Simulations were than performed in continuous time.

2.6 Sediment diagenesis model

Water quality model CE-QUAL-W2 is capable of simulating some simple processes inherent to the sediment layers of waterbodies but not in a way that provides results for methane emissions by itself. In the latest release of CE-QUAL-W2 version 4.0, the water quality model was complemented with an add-on focused only in the physicochemical processes that occur in the sediments and its interactions with the waterbody and atmosphere. This sediment-focused addition was developed for the Cumulative Environmental Management Association, specifically for predicting sediment and water quality in oil sands pit lakes but was later added as an extension of CE-QUAL-W2 (Berger & Wells, 2014). The basis for the mathematical formulation of each component of sediment diagenesis was a modelling framework developed by Di Toro & Fitzpatrick (1993) and Di Toro (2001), which calculated sediments oxygen demand and phosphorus and nitrogen release as functions of the downward flux of carbon, nitrogen and phosphorus from the water column (Zhang et al., 2015). Data input required by the sediment diagenesis model was scarce, since it requires very detailed information about the sediment bed that isn’t regularly measured and readily available in Portuguese databases. Therefore, most of the parameters use the model’s standard values instead of field measured values. 3 RESULTS 3.1 Model calibration Hydrodynamic and water quality calibration was a process required to verify that the model is in fact reproducing the processes taking place in the reservoir and showing that these are an acceptable representation of reality. In this process, model results were compared to field data from SNIRH’s database, making the necessary adjustments to the model parameters so that the obtained results match the field data gathered. Simulations were performed from 1/10/2002 (Julian day 274) to 31/12/2004 (Julian day 1096). First thing to verify is the comparison between water level and stored volume in the reservoir, granting that for the same stored volume, both the reservoir and respective bathymetric representation are in conformity. These results show that the bathymetric representation of the reservoir is well adjusted to its real counterpart and assures the possibility of a viable water level calibration (Figure 3). Water level results indicate a correct representation of the hydraulic regime of the reservoir, showing similar water level values in the entire simulation time. Water quality calibration was focused on temperature, dissolved oxygen, suspended solids, dissolved solids, phosphate, nitrite and nitrite, ammonium and alkalinity. Stratification was evident during summer

Figure 3. Water level calibration result.

Figure 4. Water temperature calibration result (model

results represented as a line and field data as points).

months, with temperature results showing a relatively steep curve with temperatures ranging between 5 ∘ C and 25 ∘ C in the water column (Figure 5).

Stratification is an important factor to develop anoxic conditions for methane production and dis solved oxygen results show a general oxygen depletion in lower depths, with zero oxygen concentration in the lower layers during summer months (Figure 6).

The remaining water quality parameters were adjusted to show results with values similar to the field data, providing a good representation of the system in general and confirming that the model is reacting as it should. Chlorophyll-a is an important part of methane production since it is a source of organic matter to be decomposed in the waterbody so it has to be calibrated with particular care (Figure 6).

3.2 Sediment diagenesis model

The sediment diagenesis model complements CE

QUAL-W2 with detailed calculations regarding Figure 5. Dissolved oxygen calibration result (model results represented as a line and field data as points). Figure 6. Chlorophyll-a calibration results (model results represented as a line and field data as points) sediment layers in waterbodies, requiring very detailed data that is yet unavailable because due to the lack of monitorization of sediments in Portuguese waterbodies. To overcome these problems, most of the values used were the standard parameters from the example documentation of the model. This approach gave results to zero emissions and non-existence of dissolved methane in the waterbody, although there were favorable conditions for its production. Various attempts were made to boost methane production by raising sediment BOD values, changing mineralization rates and decay rates for methane but all were fruitless since the final results showed only a small amount of dissolved methane that rapidly decays and disappears (Figure 7).

Figure 7. Dissolved methane concentration in the sediment layer.

Although it wasn’t possible to obtain results, this is still an ongoing project of understanding and imple menting a model that is still being developed and doesn’t represent its final version, meaning that in the future different results may appear.

4 CONCLUSIONS

Methane formation in reservoirs happen in anaero bic conditions, through the degradation of the organic matter accumulated in the benthic zone. However, not all the methane produced is sent to the atmosphere, because a part of it decays and becomes inorganic carbon. The remaining methane, the dissolved part, is released by diffusion in the air-water interface and, when the saturation concentration in the water is exceeded, bubbles star to form, which will send this exceed to the atmosphere. Consequently, a part of total methane produced is released to the atmo sphere and the rest stays in the waterbody integrating its biochemical processes.

emissions depend on the climate of the reservoir.

In tropical climates, methane emissions are usually between 50 and 150 mg $C.m^{-2} d^{-1}$, however tem pered climates have lower values, usually below 50 mg $C.m^{-2} d^{-1}$. Although Portugal doesn't have a tropical climate, emissions from Portuguese reservoirs shouldn't be ignored. Vilar reservoir, characterized by least positive water quality indicators, is a valid target for the implementation of a model that integrates the benefits of water quality modelling to the simulation of greenhouse gas emissions.

Vilar holds the necessary conditions to form methane due to the water stratification process asso ciated with periods of anoxic conditions in the hypolimnion. The model allowed to adequately repro duce the hydrodynamic behavior of the entire water body as well as the principal water quality parameters such as temperature, dissolved oxygen, suspended and dissolved solids, total phosphorus, nitrogen and chlorophyll-a. Along the simulated period, the model

J. Nganga, R. Prinn, G. Raga, M. Schulz and R. Van Dorland, 2007, Changes in Atmospheric Constituents and Radiative Forcing, Climate Change 2007: The Physical Science Basis. Contribution of Working Group I to the Fourth Assessment Report of the Intergovernmental Panel on Climate Change [Solomon, S., D. Qin, M. Manning, Z.

Chen, M. Marquis, K.B.Averyt, M.Tignor and H.L. Miller

(eds.)]. Cambridge University Press, Cambridge, United

Kingdom and New York, USA.

Gunkel, G., 2009. Hydropower – A Green Energy? Tropical

Reservoirs and Greenhouse Gas Emissions. Clean-Soil

Air Water 37: 726–734, Weinheim, Germany.

Huttunen, J. T., Nykänen, H., Turunen, J., Martikainen,

P. J., 2003, Methane emissions from natural peatlands in the northern boreal zone in Finland, Fennoscandia, Atmospheric Environment, Volume 31, Issue 1, pp 147– 151, AE International – Europe Louis, V., Kelly, C., Duchemin, E., Rudd, J. W. M. & Rosenberg, D. M., 2000, Reservoir surface as sources of greenhouse gases to the atmosphere: a global estimate. BioScience, 20, pp 766–775, Alberta, Canada. Wildi, W., 2010. Environmental Hazards of Dams and Reservoirs. Near Curriculum Natural Environmental Science 88, pp. 187–197, Versoix, Switzerland. Zhang, Z., Sun, B., Johnson, B. E., 2015, Integration of a benthic sediment diagenesis module into the two dimensional hydrodynamic and water quality model – CE-QUAL-W2. Ecological Modelling 297, pp 213–231. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

The impact of stormwater overflows on stream water quality

T. Julínek & J. Říha

Institute of Water Structures, Brno University of Technology, Czech Republic

ABSTRACT: Water quality in streams passing through urbanized areas is often harmed by discharges from point sources of pollution. One of the significant pollution sources is stormwater overflows on the sewer net work. Especially during periods with low discharge in rivers, the water released from overflows can cause high concentration of pollution to be flushed out from the sewer network. Numerical models are frequently used

to quantify the effect of such stream pollution. In this paper, the results of computer modelling of the impact of sewer network overflows on the stream water quality in the city of Brno are shown. The considered river network consists of the Svratka and Svitava Rivers, which are the main receivers in Brno, and also of two small local streams, the Leskava and Ponavka Rivers. The design rainfall event and the design scenarios, such as the location and parameters of storm retention tanks and stormwater overflows, were set up by the designer of the sewer network reconstruction project. The assessment of the impact of the pollution released from overflows on stream water quality was carried out for the present state of the sewer network and for the proposed measures to be applied to it. These proposed measures consisted mainly of structural modifications to the overflows and a proposal for new stormwater tanks. The numerical modelling of stream water quality was carried out using MIKE11 computer code.

1 INTRODUCTION

In the past, municipalities were traditionally provided with combined sewer systems. Over the decades, newly built urban areas have been appended to the existing sewer mains, causing their frequent overload ing. Various technical and environmental measures have been adopted in order to provide sustainable stormwater management (Malmqvist 2000), (Thorn dahl et al. 2015). One of the technical measures for improving the insufficient capacity of combined sew ers is to construct combined sewer overflows and stormwater retention tanks.

In the European context, the European Water Framework Directive (2000) formulates objectives concerning stream water quality. In the Czech Repub lic, the immission and emission standards for the release of both municipal and industrial effluents to the surface streams are specified in Decree 23 (2010) and Decree 416 (2010). They are expressed in terms of maximum permissible concentration limits for indi vidual water quality indicators such as BOD, COD, nitrogen compounds, etc. In practice, most perma nent effluents discharges are subject to periodic water quality checks carried out by authorized bodies. However, such limit values are not prescribed for isolated release of effluent from stormwater overflows, which is in part due to difficulty of measuring released waste water during heavy rainfall (Harmel et al. 2010).

For such effluents discharges, permissible limits have to be specified individually based on the character and quantity of the released wastewater, on the water flow and quality in the receiving stream and its environmental value, and also on the results of a cost-benefit analysis which indicates the effectiveness of the measures proposed. During the last decades, numerous studies have dealt with both municipal and industrial effluents to the surface waters, including (Ogedengbe 1984), (Parent-Raoult & Boisson 2007), (Berndtsson 2013) and others. Special attention has been paid to water quality monitoring, measurement and contamination tracking (Harmel et al. 2010), (Lee et al. 2011), (Miskewitz & Uchrin 2013). Here various pollution indicators and polluting agents have been studied, such as dissolved oxygen, BOD, COD, sediment oxygen demand, nitrogen and phosphorus compounds, temperature, fecal contamination, and many others. The appropriate tool for the assessment of the impacts of sewer overflow effluent discharges into surface streams is modelling via various techniques (Veliskova & Kohutiar 1992a), (Tetzlaff 2007), (Even et al. 2007), (Morales et al. 2015). In practice, both single models of sewer and open channel networks and coupled models including both systems are used (Arheimer & Olsson 2001). The experience of the authors of this paper is that better understanding of simulated processes is provided by modelling two separate systems, namely a sewer and an open channel system. The reason of using separate models was also easier and faster calibration of both models and more transparent data handling. Also two separate teams focused on the sewer networks and open channel hydrodynamics were solving the problems individually with higher efficiency.

Even if the applied numerical solution represents a classic approach, the contribution of the paper lies in providing information about water quality issues and standards used in the Czech Republic and in the definition and determination of input data, namely quantification of the effluents from storm water overflows. It is shown that resulting maximum concentrations of individual pollution components in the stream in case of small open channels considerably exceed the immission standards, locally more than 30 times during relatively short time. The arrangements on sewer system may improve the stream water quality in terms of maximum concentration more than 2 times especially during extreme storm events.

2 THE STUDIED OPEN CHANNEL SYSTEM

Within the comprehensive reconstruction of the sewer network in the city of Brno, the improvement of exist ing stormwater separators (overflows) and also the design of new ones are planned. The rehabilitation of sewerage also involves the design of stormwater retention tanks, which lower the peak discharges in the sewers and so decrease the released amount of polluted water to receiving rivers via stormwater separators. The purpose of the presented study was to assess the changes over time in the concentration of six wa ter quality indicators, namely biological (BOD) and chemical (COD) oxygen demand, Ammonia nitrogen (N-NH4), total Nitrogen (Nt), total Phosphorus (Pt), and suspended solids (SS) in the principal rivers in the area of the city of Brno. These rivers are the Svratka and Svitava rivers and the Leskava and Pon avka streams, along which principal sewer mains are located and equipped with combined sewer over-flows (Fig. 1). The results of the study describe the devel opment of maximum concentrations of the above mentioned polluting agents along the streams and also the total average annual mass load released into surface streams.

3 METHODS

The complex project comprised two parts, namely a study of the sewer system and a study of the river network. Within both parts, hydraulics and water quality were assessed for various hydrologic scenarios and also for proposed variants of facilities built to serve the sewer network (stormwater retention tanks, storm overflows). The project was realized in co-operation with Aquatis a.s. (JSC) consulting engineers, who were responsible for the solution of sewer network issues.They provided input values for the stream water quality model in the form of different variants of remedial measures, water discharges from separators during

Figure 1. Map of the area of interest with stormwater overflows.

the design storm event, and also the water quality of sewage water released into the streams. The study of water quality in open channels consisted of: – basic data assembly and analysis, – the design and set-up of a numerical model, numerical tests and the solution of selected variants, – evaluation of the obtained results, discussion.

3.1 Data assembly and analysis

The following data were collected for the stream water quality analysis and processed into an appropriate form for use in the numerical model: – river network topology (reach lengths, confluence locations, etc.), – the geometry of streams (cross sections) including data about structures (weirs, bed drops, bridges, water intakes, outflows and transfers, etc.), – hydrologic data (catchment area, precipitation, rainfall-runoff, landuse, annual and m-day discharges, measured concentrations of individual compounds), – location of pollution sources, stormwater overflows, outlets from industrial facilities and a waste water treatment plant, – data regarding time series of waste-water discharges from individual separators, and the water quality of the released waste water. The data for the catchment, topology and geometry of open channels were obtained from archival sources

Table 1. Retention tank variants, volume in [m^3].

Sewer River Bank V1 V2 V3 V4

A Svratka R 2000 5000 5000 5000

A Svratka R 2500 5500 5500 5500

B Svratka L 5000 7000 8000 5500

D Svitava R 10000 20000 30000 20000

E Svitava L 2500 5000 4000 5000

E Svitava L 2000 5000 2000 5000

E Svitava L 500 500

E Svitava L 1000 1000

E Svitava L 500 500

Comment: R – right bank, L – left bank, V – variant

Table 2. Summary of remedial measures at overflows, variant V2.

Sewer	River	existing	new	modified	removed
A	Svratka	11	3	3	8
B	Svratka	12	2	0	0
C	Ponavka	0	1	0	0
D	Svitava	12	2	3	1
E	Svitava	15	4	7	5

owned by the Morava River Board Administration, while hydrological data were provided by the Czech Hydrometeorological Institute.

The data for the proposed variants (i.e. the location, arrangement and hydraulic parameters of retention tanks and stormwater overflows) were provided by

Aquatis a.s. (JSC) consulting engineers, who carried out numerical simulations of the rainfall-run-off pro cess and the hydrodynamics of the sewer network for each variant. The output of numerical modelling includes (among other things) the amount of wastew ater (discharge) released to the streams for each variant.The variants of the proposed stormwater reten tion tanks with corresponding volumes are shown in Table 1. The analysis included a comparison with vari ant V0 – the present state of the sewer (not mentioned in Table 1).

Cost-benefit analysis showed that variant V2 had the best efficiency, and therefore the following sum mary of remedial activities on stormwater overflows is related to this variant.

There are 50 existing stormwater overflows in the Brno sewer system.The proposal was to add another 12 overflows along the main sewer collectors (named A, B, C, D, E) in order to modify 13 existing overflows for the purpose of improving their hydraulic function, and also to remove 14 existing unsatisfactory overflows. The summary of overflows for the resulting variant V2 is shown in Table 2.

The concentration of individual components of released sewage water had to be derived from the observed data taken from the sewer network during

extreme rainfall. However the regular water quality Table 3. Concentrations [mg/l] of released sewage water from strorm water overflows located along individual rivers. Present state Improved Svratka Svitava Svratka Svitava BOD 5 117 92.7 95/51 62/43 COD 508 288 256/111 192/134 N-NH 4 10.6 17.1 5.1/2.2 10.8/7.6 Nt 26.4 31.6 19.6/10.4 20/14 Pt 4.3 3.1 4.0/2.2 2.2/1.5 SS 921 435 605/301 290/203 Note: Values behind the slash stands for the overflows with retention tank. Table 4. Concentrations [mg/l] in surface streams during “no pollution” period. BOD 5 COD N-NH 4 N t P c SS 4 22 0.3 5.5 0.4 10 monitoring at sewer systems is still not a standard in the Czech Republic. Therefore three sampling “campaigns” were organized by the South-Moravian Water Supply Administration to gather realistic pollution data from the sewers during storm events in various parts of the city of Brno. The campaigns showed quite unpredictable results with no typical trend or behaviour. In several cases, probably due to sudden scour or flush out of pollution from sewerage network the concentrations increased about than 10 times instantaneously with the duration of few minutes. Moreover, the change in concentration occurred for different indicators in different moments not corresponding to changes in discharge. The data obtained were therefore compared with generally used wastewater data published in the literature and experienced in other localities. Finally, a constant concentration (over time) of released sewage water was agreed by the Administration’s steering committee (Table 3). The discharges in streams during “no pollution” periods when the storm overflows were not in operation were derived from hydrological data for two scenarios, namely annual average discharge and 270day discharge. Based on the long-term water quality sampling the concentration in streams was considered as same for both discharge scenarios. The relevant values are shown in Tables 4 and 5. All of the gathered spatial information and data were processed using the ARC VIEW geographic information system (GIS). The GIS was linked to the numerical model and also enabled final integrated data analysis and processing. 3.2 Numerical modelling A numerical model was applied to evaluate the present state of water quality in the streams and to assess the

Table 5. Discharges [m 3 /s] in surface streams during “no

pollution” period.

River Reach (Fig. 1)	Q_a	Q_{270}
Svratka upstream of the Ponavka	7.16	2.95
Ponavka	0.40	0.20
Svratka between the Ponavka and Leskava	7.56	3.15
Svratka between the Leskava and Svitava	7.63	3.18
Svratka between the Svitava and WWTP	12.10	5.30
Svratka downstream of WWTP	13.16	5.71
Svitava km 11.0 to km 6.47	4.87	2.32
Svitava km 6.47 to the Svratka	4.47	2.12
Leskava	0.07	0.03

WWTP – wastewater treatment plant

effects of proposed stream water pollution improvement scenarios via structures on the principal sewer mains. A numerical model of stream water quality consisted of hydrodynamic and pollution transport modules. The effect of transversal mixing could be neglected as the sewer separators are in most cases located on both banks of the streams, and also due to the relatively small width of the streams (Veliskova & Kohutiar 1992b). A one-dimensional convection-dispersion model was applied in order to obtain a solution:

with concentration c, mean stream velocity u, coefficient of longitudinal dispersion D_L, constitutive changes f and pollution sources R. All variables are a function of time (t) and river stationing (x).

Mean stream velocity u was determined by the 1D

transient hydraulic model (a module within MIKE11

software).

Dispersion coefficient D L was expressed as a prod

uct of stream velocity u and constant α (sometimes

called dispersivity):

The dispersivity for individual river reaches was taken

from the results of previous extensive research carried

out in the area of interest (Rˇíha et al. 2001).The disper

sivity values used in modelling are shown in Table 5.

These values correspond to hydrodynamic dispersion

D L = 5–8 m 2 /s.

Pollution sources R were specified as known mass

flow during the release of sewage water from the

stormwater overflows. It was calculated as the prod

uct of constant concentration (Table 3) and time

dependent discharge released from the sewer sys

tem via stormwater overflows (Fig. 2) determined

by the hydraulic model of the sewer network set up

with the use of MOUSE software byAquatis a.s. (JSC).

As the entire event at the related streams was relatively

short (a few hours only) the system was considered to

be conservative with no constitutive changes (f = 0). Table
6. Dispersivity [m] used for individual river reaches.
Svratka * Svratka ** Svitava Leskava 11.5 10 8 6 Note:
Svratka * – is the Svratka River upstream of the Svitava

River, Svratka ** – is the Svratka River downstream of the junction with the Svitava River. Figure 2. The hydrographs at the stormwater overflows. At the hydrodynamic module the initial condition and also upper boundary conditions expressed two scenarios represented by the average annual discharges according the Table 5. The upper boundary conditions were introduced in the stream profiles upstream of the city of Brno which were not influenced by sewer overflows. The downstream boundary condition was set up as the rating curve at the downstream profile of the Svratka River at Zidlochovice located far enough from the downstream profile of interest. For the water quality model the upstream boundary conditions were specified as known concentrations in the river profiles beyond the city of Brno. The values from Table 4 were used. The downstream boundary condition (3) was set as a zero concentration gradient for the downstream profile at Zidlochovice mentioned above: The initial condition was represented by a constant concentration in the streams during the "no pollution" period as shown in Table 4. 3.3 Results of simulations The main objective of the study was to assess the effect of measures applied to the sewer system on the water quality in streams in the urban area of the city of Brno.Two scenarios were resolved, namely the present state and a proposed set of improvements involving the reconstruction of overflows and rainfall storage tanks, increasing the capacity of sewer mains, and the comprehensive intensification of the local wastewater treatment plant.

Table 7. Maximum calculated concentrations of pollutants

in [mg/l] and multiple of immission limits (IL) – present state

at discharge Q_a .

Indic.	Maximum concentration in [mg/l]/multiple of immission limit				IL
	Svratka *	Svratka **	Svitava	Leskava	
BOD_5	116/19	65/11	62/10	4.0/0.7	6
COD	293/8.4	193/5.5	195/5.6	22.0/0.6	35
$N\text{-}NH_4$	6.0/12.0	5.6/11.1	11.1/22.3	0.3/0.6	0.5
N_t	26/3.2	16/2.0	22/2.8	5.5/0.7	8

P t	4.8/31.9	2.5/16.4	2.1/14.1	0.4/2.7	0.15
SS	662/27	351/16.4	281/14.1	10/0.4	25

Note: Svratka * at km 44.11, Svratka ** at km 40.19, the River Svitava at km 11.2 and the Leskava River at km 1.15. IL – immission limit.

Table 8. Maximum calculated concentrations of pollutants in [mg/l] and multiples of the immission limit – after proposed improvements at discharge Q a .

Maximum concentration [mg/l]/multiples of immission limit (IL)

Indic.	Svratka *	Svratka **	Svitava	Leskava	IL
BOD	64.7/10.8	35.3/5.9	30.3/5.1	90/15	6
COD	177.5/5.1	105.3/3.0	98/2.8	242/6.9	35
N-NH 4	3.5/7.0	3.0/6.1	5.1/10.2	4.8/9.6	0.5
N t	14.8/1.9	10.0/1.3	11.1/1.4	18.7/2.3	8
P t	2.8/18.5	1.5/9.8	1.1/7.5	3.8/25.2	0.15
SS	405/16.2	191.0/7.6	137/5.5	567/22.7	25

Note: * , ** see Table 7.

The results of modelling showed that:

- at present, the stream water quality during extreme rainfall events in the Brno area exceeds the limits prescribed by immission water quality standards by more than 10 times (in some cases, e.g. the Svitava river and N-NH4, the limits were exceeded by more than 20 times),

– the improvements mentioned above will consider

ably decrease the amount of pollution released. At

the most critical reaches along the Svratka and Svi

tava rivers the concentration of pollutants will be

reduced by approx. 50%. However, stream water

quality standards (immission limits) will still be met

during extreme events,

– in the case of a simulated extreme event the water

quality standards in the analyzed rivers will be

violated for a period of more than two hours by

approximately 3 times greater concentrations than

those accepted by the Decree 23 (2010). Smaller

water courses will suffer extreme exceedance of

the accepted value (by 20 times), but for a shorter

period.

The results of the calculation are summarized in

Tables 7 and 8 for the discharge Q a , in Tables 9 and

10 for the discharge Q 270 . The maximum calculated Table 9. Maximum calculated concentrations of pollutants in [mg/l] and multiple of immission limits (IL) – present state at discharge Q 270 . Maximum concentration in [mg/l] / multiple of immission limit Indic. Svratka * Svratka ** Svitava Leskava IL BOD 5 134.0/22.3 71.6/11.9 70.3/11.7 4.0/0.7 6 COD 368.3/10.5 217.4/6.2 221.9/6.3 22.0/0.6 35 N-NH 4 7.6/15.2 6.63/13.3 12.78/25.6 0.30/0.6 0.5 N t 28.9/3.6 17.0/2.1 24.6/3.1 5.5/0.7 8 P t 5.5/36.7 2.6/17.2 2.4/15.8 0.5/3.3 0.15 SS 755.3/30.2 376.5/15.1 322.1/12.9 10.0/0.4 25 Note: * . ** see Table 7. Table 10. Maximum calculated concentrations of pollutants in [mg/l] and multiples of the immission limit – after proposed

improvements at discharge Q 270 . Maximum concentration [mg/l] / multiples of immission limit (IL) Indic. Svratka * Svratka ** Svitava Leskava IL BOD 77.8/13.0 41.0/6.8 34.2/5.7 92.5/15.4 6 COD 210.8/6.0 120.6/3.4 110.8/3.2 248.6/7.1 35 N-NH 4 4.16/8.3 3.17/6.3 5.78/11.6 4.95/9.9 0.5 N t 18.8/2.4 10.7/1.3 13/1.6 19.1/2.4 8 P t 3.29/21.9 1.65/11.0 1.13/7.5 3.78/25.2 0.15 SS 489.8/19.6 222.5/8.9 155.9/6.2 586.4/23.5 25 Note: * . ** see Table 7. concentrations of pollutants observed along the individual river reaches are mentioned and compared with immission limits prescribed by Decree 23 (2010). As an example, a comparison of the modelling results in terms of BOD concentrations in the Svratka and Svitava Rivers for the discharge Q a is shown in Figures 3 and 4. The stream junctions and stormwater overflow locations are shown in Figure 1. 4 CONCLUSIONS The study assessed the effect on the sewer system of proposed measures which aim to decrease the concentration of pollution released into receiving rivers. The results obtained from the model enabled a comparison of the present state with variant V2 of the proposed set of improvements. The accuracy and reliability of model solutions greatly depends on the accuracy and credibility of input data. Here it must be noted that only limited data on the water quality in the sewer system and also in the investigated rivers during storm events were available for reliable model calibration. The model solution results show that the measures proposed for the sewer system (retention tanks, overflows, etc.) will reduce the peak concentrations of

Figure 3. Comparison of the present (V0) and proposed (V2) state for the Svratka River – maximum BOD concentration

envelope at the discharge Q a .

Figure 4. Comparison of the present (V0) and proposed (V2) state for the Svitava River – maximum BOD concentration

envelope at the discharge Q a .

individual monitored indicators of water quality by

45 to 90% of those recorded for the present state.

The increase in concentration of water quality indi

cators will occur at the upper part of the Svratka River (stationing 45.5–50.0, see Fig. 3) and the Leskava River, where the new overflows are proposed. The maximum concentrations shown in Tables 7 and 8 and in Figures 3 and

4 represent concentration envelopes during storm events when wastewater dis charges from the overflows. The occurrence time of the concentration peaks varies from 10 minutes in the upper river reaches up to several hours at the reaches close to the junction of the Svitava and Svratka rivers. Thus the immission standards in the streams are not fulfilled during the extreme storm events. Emission standards are practically impossible to be monitored at the storm water overflows. The limits and thresh olds given by Decree 416 (2010) were significantly exceeded. However, Decree 416 cannot be directly applied for spills from overflows and water quality in rivers during rainfall as the Decree was originally only intended for permanent pollution discharge. Limits must be set on the basis of the ecological condition of each individual watercourse and the agreement of spe cialists (sewer system managers, ichthyologists, water quality engineers, and others).

The numerical modelling of the effects of different proposed measures is a good design optimization tool. In general two criteria can be applied. These are the resulting maximum instant concentration of pollution in surface water in mg/l, and the overall mass of pollu tion in t/year, or during storm events. These data, when linked to the investment cost of the considered mea

sures, can provide basic information for the decision

making process.

ACKNOWLEDGEMENT

This paper has been prepared under projects

No. LO1408 AdMaS UP Advanced Materials, Struc

tures and Technologies and FAST-S-15-2841 Over

flows under specific hydraulic conditions

Arheimer, B. & Olsson, J. 2001. Integration and Coupling

of Hydrological Models with Water Quality Models:

Applications in Europe. Swedish Meteorological and

Hydrological Institute, Sweden, 53 p.

Berndtsson, J.C. 2014. Storm water quality of first flush urban

runoff in relation to different traffic characteristics. Urban

Water Journal, Volume 11, Issue 4, 2014, 284–296 p.

Booth, B., Crosier, S., Clark, J. & MacDonald A. 2002.

“Building a Geodatabase”, ESRI, New York, USA, 460

p.

Booth, B. & Lamirand J. 2003 Using Image Analysis for

ArcGIS, EDAS, Atlanta, USA, 207 p.

Decree23. 2010. Decree No. 23/2010 Coll. on indicators

and values of permissible pollution of surface water and

wastewater, details of the permit to discharge wastewa

ter into surface water and into sewerage systems and in sensitive areas. (Czech national legislation).

Decree416. 2010. Decree no. 416/2010 Coll. on indicators

and values of permissible pollution of waste water and

the details of the permit to discharge waste water into

groundwater. (Czech national legislation).

Directive 2000/60/EC of the European Parliament and of the

Council establishing a framework for Community action

in the field of water policy. Even, S., Mouchel, J-M., Servais, P., Flipo, N, Poulin, M., Blanc, S., Chabanel, M. & Paffoni, C. 2007. Modelling the impacts of Combined Sewer Overflows on the river Seine water quality. Sci Total Environ. 2007 Apr 1;375 (1–3):140–51 p. EWFD. 2000. Directive 2000/60/EC of the European Parliament and of the Council establishing a framework for Community action in the field of water policy Harmel, R. D., Slade, R. M. & Haney, R. L. 2010. Impact of Sampling Techniques on Measured Stormwater Quality Data for Small Streams. J. Environ. Qual. 39:1734–1742 p. Lee, S., Lee, J. & Kim, M. 2012.The influence of storm-water sewer overflows on stream water quality and source tracking of fecal contamination. KSCE J. of Civil Engineering (2012) 16(1):39–44 p. Malmqvist, P.A. 2000. Sustainable storm water management -some Swedish experiences. Journal of Environmental Science and Health Part A A35(8):1251–1266. Miskewitz, R.J. & Uchrin, Ch.G. 2013. In-Stream Dissolved Oxygen Impacts and Sediment Oxygen Demand Resulting from Combined Sewer Overflow Discharges. In: Journal of Environmental Engineering 139(10):1307, 2013. Morales, V., Mier, J. & Garcia, M. 2015. “Innovative modelling framework for combined sewer overflows prediction.” Urban Water Journal, 10.1080/1573062X.2015. 1057183, 1–15. Ogedengbe, O., Fapohunda, H. O. & Gotau, M. B. 1984. Studies on a Selected Industrial Effluent and the Receiving Stream. In: International Journal of Environmental Studies 23(3-4):229–234. Parent-Raoult, Ch. & Boisson, JC. 2007. Impacts des rejets urbains de temps de pluie (RUTP) sur les milieux aquatiques: État des connaissances. Revue des sciences de l’eau / Journal of Water Science, vol. 20, n ◦ 2, 2007, p. 229–239. Rˇíha, J. et al. 2001. The methods of mathematical modelling stream water quality. Final Report of Grant project GACR No. 103/99/0456. Water Structures Institute, FCE BTU Brno. Tetzlaff, D., Uhlenbrook, S., Grottker, C. & Leibundgut, CH. 2005. Hydrological assessment of flow dynamic changes by storm sewer overflows and combined sewer overflows on an event-scale in an urban river. Urban Water Journal 2(4). 201–214 p. Thorndahl, S.L., Schaarup-Jensen, K. & Rasmussen, M.R. 2015. On Hydraulic and Pollution Effects of Converting Combined Sewer Catchments to Separate Sewer Catchments. In: Urban Water Journal, Vol. 12, No. 2, 2015, 120–130 p. Veliskova, Y. & Kohutiar, J. 1992a. The determination of transverse mixing coefficient in wide and shallow rivers. [Urcenie koeficientov priecneho zmiesavania v sirokych plytkych tokoch]. In: Vodohospodarsky Casopis/Journal for Hydrology & Hydromechanics. Volume 40, Issue 6, 1992, p. 506–516. Veliskova, Y. & Kohutiar, J. 1992b. Two-dimensional model of dispersion in natural channels. [K dvojrozmernemu modelovaniu disperzie v prirodzenych korytach]. In: Vodohospodarsky Casopis/Journal for Hydrology & Hydromechanics. Volume 40, Issue 5, 1992, p. 409–424. Vogel, J.R., Frankforter, J.D., Rus, D.L., Hobza, C.M. & Moser, M.T. 2009. Water quality of combined sewer overflows, stormwater and streams, Omaha, Nebraska, 2006–07: U.S. Geological Survey Scientific Investigations Report 2009–5175, 152 p. plus appendixes. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Groundwater quality of Feriana-Skhirat in Central Tunisia and its

sustainability for agriculture and drinking purposes

I. Hassen

Modeling Hydraulic and Environmental Laboratory, National Engineering School of Tunis,

University of Tunis El Manar, Tunisia

F. Hamzaoui-Azaza

Research unit of Geochemistry and environmental Geology, Faculty of Science of Tunis, University

of Tunis El Manar, Tunisia

R. Bouhlila

Modeling Hydraulic and Environmental Laboratory, National Engineering School of Tunis, University of Tunis El Manar, Tunisia

ABSTRACT: Feriana-Skhirat is a deep groundwater aquifer, located in Central Tunisia. Groundwater exploited for Miocene aquifer is the main water resource for the region. The system is composed of unconfined Miocene sandstone with a thickness about 200 m. The aquifer lies deep under a thick layer of marl and clay of Upper Miocene exceeding the thickness of 400 m. Sampling surveys were undertaken in January-February 2014 from 13 wells and concentration of different physico-chemical parameters (TDS, Na^+, Ca^{2+}, Mg^{2+}, HCO_3^-, SO_4^{2-}, Cl^-, pH and temperature) were considered to determine the origin of mineralization and the suitability of groundwater for irrigation and drinking consumption. The hydrochemical data interpretation reveals that the mineralization is mainly regulated by water/rock interaction through mineral dissolution and ions exchange processes. The water facies gradually changes from SO_4^{2-}-Ca^{2+}-Mg^{2+} to SO_4^{2}–Na^+ facies. All the major ions fall in the permissible range of the WHO (2008). On the basis of the Water Quality Index WQI, Feriana-Skhirat aquifer has been classified as “Excellent Water”. Sustainability of groundwater for irrigation based on the Wilcox and Richard diagrams, indicating that groundwater serves good irrigation quality. Groundwater samples fall within the (C3-S1) field, indicating good irrigation quality. In the C4-S2 field, two samples were found; indicating very high salinity and medium alkalinity hazard. Water with such characteristic may be used for plants tolerating high salinity.

1 INTRODCUTION

Groundwater is an important source of water supply throughout the world. In Central Tunisia, groundwater is the primary water source for drinking, agriculture and industrial purposes. The combination of popula tion growth, economic and agriculture development leads to the overexploitation of the water resources and to the deterioration of its quality.

Localized in the government of Kasserine in Central Tunisia, Feriana-Skhirat is among the most impor tant groundwater in the region. Characterized by an arid climate, Feriana-Skhirat represents the primary source of water for several uses and provides the main groundwater potential for water supply in the study area.

Because of the intensive agriculture activities throughout the region, the groundwater demand is currently being increased. This fact leads to the over pumping and by the way to overexploitation and deterioration of water quality in these arid regions during the last decades (Hamzaoui-Azaza et al, 2011). Therefore, water quality issues need to be given greater attention in arid and semi arid regions. In order to avoid such problems, understanding the groundwater geochemistry, origin and evolution is of high importance. Plenty of scientists in the last decades focus their studies on understanding and investigating the water geochemistry and the hydrochemical processes of groundwater (Hamzaoui Azaza et al., 2012; Ameur et al., 2015, Li et al., 2012; 2013; Wu et al., 2014; Kumar et al., 2004, 2007, 2015; Hassen et al., 2016). To be a part of such studies, this paper deals with the study of the main geochemical processes controlling Feriana-Skhirat groundwater mineralization and water quality for promoting agriculture sustainability and drinking consumption. To attempt these purposes, several methods were used, such as conventional classification techniques, correlation parameters and geochemical modelling. To evaluate groundwater

Figure 1. Study area

sustainability for drinking purposes in Feriana-Skhirat aquifer, the Water Quality Index (WQI) was calculated and a comparison with the WHO (2008) standards was established. Besides, the groundwater irrigation was assessed using numerous diagrams such as Wilcox and Richards's diagrams. Such diagrams lead to delineate regions where groundwater is suitable or unsuitable for irrigation purpose.

2 STUDY AREA

Located in South-Western of Kasserine (Central Tunisia), the deep aquifer of Feriana-Skhirat is one of the most important groundwater in Kasserine aquifer system. The study area is extended from the small syn clinal of Feriana to the Synclinal of Skhiart till the Algerian border. It is bordered by mountains such as Dj Sarraguia, Douira, Goubel, El Kebir and Feriana (fig. 1).

Feriana-Skhirat is influenced by an arid climate with a mean annual precipitation of 289 mm, mean annual temperature of 17 ∘ C and potential evapo transpiration of about 2130 mm/year. Surface water resources are represented by wadi Esaboun wadi el

Kiss and Bouhaya and Wadi Goubel and Wedi safsaf

in the Algerian border. The water flows are not peren

nial and occur only by flash floods in rainy seasons

(Yangui et al.2012).

Geologically, the mean geological features of this

study area are:

a) Upper Cretaceous: composed of limestone outcrop

ping in the study area, represented by the Campa

nian and Maestrichtian known as Abiod Formation

in Jb Sarraguia, Goubel, El Kebir and Feriana. This layer may represent in many regions a potential aquifer. b) Upper Miocene: Composed mainly of clay and marl inter-bedded with thin layers of sand. The marl of the Upper Miocene sediments thickens up to 400 m in the synclinal of Skhirat. Miocene: deposits are composed of sand and sandstone known as Vindobonian formation. The aquifer deposits are characterized by good permeability and range thickness between 200 and 400 m. This layer represents the main potential aquifer in Kasserine and Central Tunisia. Tectonically, the Feriana-Skhirat aquifer is delimitated by two main vertical faultsA2 and C2, separating Feriana-Skhirat aquifer from Majel Bel Abbes. Also in the same area we can found the Fault B2 leading to the formation of Ain Kiss spring. The following description of the geological framework for Feriana-Skhirat is summarized in various geological and hydrogeological reports (Berkaloff 1931, Degalier 1952, Khanfir 1980; Hassen 2013, Hassen 2014) as well as from borehole lithology logs of drilled water wells in the region. Groundwater recharge in Feriana-Skhirat results from direct rainfall infiltration and through the wedis. It may also occur from the Miocene sandstone outcrop in the Algerian border. Hydrogeologically, the aquifer geometry is unconfined in the west of the small synclinal of Feriana and in the West of Gour Essouan near the Algerian Border. Except these zones, the Miocene sandstone lies deep under a thick layer of clay and marl of Upper Miocene. The thickness of this deep aquifer reaches 400 m. Feriana-skhirat groundwater is exploited by deep wells for irrigation, drinking and also industrial activities. The total output of the region of Kasserine was estimated of about 72.94 Mm 3 in 2004 that registered in 2003 was 64.12 Mm^3 that lead to the overexploitation of this aquifers system (DGRE General Direction of Water Resources, 2004). 3 METHODOLOGY 3.1 Sample collection and analytical techniques Feriana-Skhirat groundwater has been studied in terms of physico-chemical parameters (Tab. 1). These parameters were measured in February 2014 for 13 deep wells used for drinking and irrigation (fig. 1). Protocol for sample collection and preservation followed the Standard Methods of APHPA (APHPA 1995). Fieldwork included measurement of temperature, pH, salinity and electrical conductivity that can require representation values of ambient aquifer while the chemical analyses including majors ions and some minors elements were conducted in the laboratory of the Centre of Hydrogeology and Geothermic (CHYN) of the University of Neuchatel (Switzerland) by liquidion chromatography. The ionic charge balance of each sample was within +/-5%.

Table 1. Analytical results for physic-chemical parameters and major ions

Param (mg/l)	Min	Max	Mean	WHO	w_i	W_i
TDS	579.6	1821.37	858.21	1500	5	0.147
pH	6.63	6.63	7.36	6.5-8.5		
T	12.2	20.2	17.25	-		
Na	48.89	371.68	97.15	200	3	0.088
Ca	68.21	163.57	97.09	200	3	0.088
K	2.13	5.27	3.24	30	3	0.088
Mg	25.32	55.9	37.16	200	3	0.088
HCO_3	213.5	305	247.22	380	3	0.088
Cl	27.051	153.48	83.41	250	2	0.059
SO_4	141.76	844.52	292.92	400	2	0.059
NO_3	9.47	41.7	23.4	45	5	0.147

3.2 Saturation Index

PHREEQC (Parkhurst and Appelo 1999) is a geo

chemical modeling software used in order to study

the saturation state of the water with respect

to halite(NaCl), gypsum ($CaSO_4 H_2O$), anhydrite

($CaSO_4$), calcite ($CaCO_3$) and dolomite $CaMg(CO_3)_2$.

It allows the computation of the saturation index for

each mineral; hence the determination of the main

dissolution or precipitation process controlling the

groundwater chemistry (Hassen et al, 2016). The

Saturation Indices are expressed as (Aghazadeh and

Mogaddam, 2010):

Where: SI = Saturation Index,

IAP = ion activity product

K t: = the equilibrium solubility constant

3.3 Water quality index

To evaluate the quality of water and its suitability for

drinking usage, Water Quality Index (WQI) meant

to be a useful and decent method for such purpose

(Hamzaoui-Azaza et al., 2013; Hassen et al., 2016).

This index may be defined as a mathematical method

used to transform large quantities of water quality data

into a single number which represents the water quality

level.

WQI was calculated at three steps; in the first one,

a total of ten chemical parameters ((TDS), Na^+, K^+,

Ca^{2+}, Mg^{2+}, Cl^-, SO_4^{2-}, HCO_3^- and NO_3^-) has been

assigned a weight (w_i) according to its relative impor

tance in the quality of water for drinking purposes.

Generally, the highest weight is assigned to the element

with high concentration owing to their major signif

icance in water quality assessment and their health

implications while the minimum weight is assigned to

the elements with the least importance in water quality

assessment (Serinivasamoorthy et al., 2008; Yiadana

et al., 2010a; Varol and Davaraz, 2014). The second step consists of the computation of the relative weight (W_i) which is calculated from the following equation: where wi is the weight of each parameter, n is the number of parameters In the third step, a quality rating scale (qi) from each parameter is computed from: where q_i: the quality rating, C_i: the concentration of the chemical parameter in the water sample (mg/L), S_i: the drinking water standard for each chemical parameter to the guidelines of the WHO (2008) (mg/L). The sub index SI_i of the parameter and the WQI were respectively calculated from these equations: 3.4 Sustainability of groundwater for irrigation The suitability of groundwater for irrigation purposes depends on the effect of mineral constituents of water on both plants and soil. The main criteria to assess the irrigation water quality are: total salt concentration as measured by EC, relative proportions of Na^+ as expressed by %Na, PI, SAR and boron (Ramesh and Elango, 2011). In this study, sodium percentage (%Na) and sodium adsorption ratio (SAR) was performed. High sodium concentration is a threat for plant growth and its percentage in soil is critical for determining groundwater suitability for irrigation purposes (Chung et al., 2014).This percentage can be calculated using the formula (Wilcox, 1955) given below, where the concentrations are expressed in meq/L. As for sodium adsorption ratio (SAR), it is defined as a measure of alkali and sodium hazard to crops. It is a relevant criterion for deciding the suitability of groundwater for irrigation (Aghazadeh and Mogaddam, 2010). SAR is defined by (Karanth, 1987) Where all the concentrations are in meq/L. 4 RESULTS AND

DISCUSSION 4.1 Hydrochemical characteristics Table 1 represents the statistical summary of the hydrochemical parameters of Feriana-Skhirat: The salinity content varies from 586 mg/l to 1821 mg/l with a

mean value of 858.2 mg/l. The potassium concentra

tion has a low and homogenous concentration ranged

between 2.13 mg/l and 5.27 mg/l. This low level of

potassium may be explained by its tendency to fix

clay minerals existing in the Upper Miocene layer

and participating in the formation of secondary miner

als (Hamzaoui-Azaza et al.2012). As for the calcium

and magnesium concentrations, it ranged respectively

from 68 to 164 mg/l and from 25.3 to 56 mg/l. Water

alkalinity of Feriana-Skhirat aquifer is the only prod

uct of bicarbonate ions, given that pH values are almost

near to neutral. The bicarbonate ion concentrations

ranged from 213.5 to 305 mg/l. The bicarbonate con

centration in groundwater is derived from carbonate

weathering. The chloride and sulfates concentrations

have mean values of 83.4 and 293 mg/l respectively.

The sequence of abundance of cations and anions

follow the following order: SO 4 > HCO 3 > Cl and

Na > Ca > Mg > K.

4.2 Geochemical facies

Piper tri-linear diagrams have been widely used since

it is helpful in understanding the hydro-geochemical

regime of a study area (Li et al., 2013). The Piper classification of Feriana-Skhirat aquifer in fig. 2 distinguished two groups: The first one has SO_4^{2-} Ca^{2+}-Mg^{2+} facies and the second one has SO_4^{2-}-Na^{+} facies. The SO_4^{2-}-Ca^{2+}-Mg^{2+} water type results mainly from carbonate minerals precipitation and dissolution of gypsum minerals which are relatively abundant within the Upper Miocene sequences (Ouda 2000; Yangui et al. 2012; 2012). While, the SO_4^{2-}-Na^{+} type water, represented by FS4 and FS11, is controlled by rock water interaction. This facies is highly mineral ized comparing to the other wells and results from the substitute of calcium by sodium through cation exchange.

4.3 Source of major ions and hydrochemical evolution

Understanding the geochemical evolution of the major ions in Feriana-Skhirat groundwater was studied in the base of correlation analysis and ions ratios. In addition, variation in the saturation indices of major minerals along flow paths was also investigated.

4.3.1 Correlation parameters-Ions ratios and Ion exchange

The correlation matrix represented in Tab. 2 describes the inter relationship between variables (13 wells) and the results for 8 hydrochemical parameters: A very high correlation exists between TDS and Na^{+}, Mg^{2+}, Ca^{2+}, Cl^{-} and SO_4^{2-} indicating that mineral ization is controlled by these referred ions. Natural concentrations of sulfates may be enriched by weath ering of sulfate minerals, such as gypsum and anhy drite. In this present study, a high correlations exists between Na^{+} -SO_4^{2-} , Mg^{2+} -SO_4^{2-} and Ca^{2+} -SO_4^{2-} , ($R^2 > 0.8$) indicating the contribution of evaporate minerals (gypsum) presents in the aquifer. Also, a

high positive correlation exists between Ca^{2+} and Figure 2. Piper diagram. Mg^{2+} ($R^2 = 0.95$) indicating that these ions derivate from the same origin which may be the weathering of dolomite and limestone (Subramani et al., 2005; Hamzaoui-Azaza et al. 2013). Furthermore, A value of $Mg^{2+}/(Ca^{2+} + Mg^{2+}) < 0.5$ for all the groundwater samples indicates that these ions display a similar geochemical behaviour and have a common natural origin which may be dolomite and limestone weathering (Hounslow 1995). The possible source of sodium and chloride concentration in groundwater would be due to the dissolution of rock salts and weathering of sodium-bearing minerals (Krishna Kumar et al., 2009, Hamzaoui-Azaza et al. 2013). If the Na/Cl ratio is approximately equal to 1, then the halite dissolution is the main source of sodium. Furthermore, if the Na/Cl ratio greater than 1, it reflects generally a release of sodium from silicate weathering (Meyback, 1987). In the case of Na/Cl ratio is less than 1, the reduction of Na concentration may be due to ion exchange process. The most groundwater samples of FerianaSkhirat (8 samples) have a Na^{+}/Cl^{-} molar ratio less than 1 suggesting that the chemical evolution of this groundwater is controlled by ion exchange process between groundwater and the clay layer present in the Upper Miocene horizon. As for sodium ion, it can be released into solution by cation exchange with calcium that is attached to the surface of newly formed clay minerals (HamzaouiAzaza et al. 2013). In order to argue that the ion exchange process affects the hydrochemistry of Feriana-Skhirat aquifer, the ($Ca^{2+} + Mg^{2+}$) vs ($HCO_3^{-} + SO_4^{2-}$) scatter diagram in Fig 3was plotted. It shows that the groundwater samples fall in the left side among the line 1:1 indicating that ion exchange reaction may be the main process controlling the geochemistry of Feriana-Skhirat aquifer (Kumar 2004; Vungopal, 2008; Belkhiri et al. 2012;

Hamzaoui-Azaza et al. 2012) 4.3.2 Saturation Index The influence of water-rock interactions on groundwater hydrochemical characteristics is reflected by the extent of mineral dissolution and precipitation, which

Table 2. Correlation matrix of the hydrochemical

parameters	TDS	Na	K	Mg	Ca	HCO_3	Cl	SO_4
TDS	0							
Na	0.95	0						
K	0.23	0.05	0					
Mg	0.90	0.7	0.3	0				
Ca	0.85	0.6	0.5	0.9	0			
HCO_3	0.43	0.5	0.1	0.3	0.1	0		
Cl	0.62	0.4	0.5	0.8	0.8	0.027	0	
SO_4	0.99	0.9	0.2	0.8	0.8	0.36	0.5	0

Figure 3. (a) Plot (Ca^{2+} +Mg^{2+}) against (HCO_3^- + SO_4^{2-})

in meq/l, (b) Plot [(Na^+ + K^+)-Cl^-] against

[(Ca^{2+} +Mg^{2+})-(SO_4^{2-} +HCO_3^-)] in meq/l.

is in turn determined by the saturation index (SI) (Li et al., 2013)

In the present study, the water samples of Feriana Skhirat are saturated/under saturated with respect to Dolomite and Calcite ($-1.7 <$ SI (Dolomite) < 1.3 and $-0.8 <$ SI (Calcite) < 0.7). With such results, carbonate precipitation may occur systematically for the majority of groundwater samples. Consequently, the precipi tation of these minerals causes equilibrium in Ca^{2+}

concentrations and leads to the dissolution of evap

orate materials (Gypsum and anhydrite) (Hamzaoui

Azaza et al., 2011; Hassen et al, 2016). Otherwise,

there is an under saturation state with Gypsum,

Halite and Anhydrite (−1.5 < SI (Gypsum) <−0.73 and

−7.4 < SI (Halite) <−6.08 and −1.75 < SI (Anhydrite) <

−0.9) indicating dissolution of these evaporate min

erals. Therefore, the soluble component for these

evaporate minerals Na^+, Cl^-, Ca^{2+} and SO_4^{2-} con

centrations are not limited by mineral equilibrium

(Belkinri, 2012).

4.4 Assessment of groundwater for Irrigation purposes and drinking consumption

Irrigation : Groundwater in Feriana-Skhirat region is

used for irrigation and drinking purposes. To evalu

ate the irrigation suitability of groundwater, Sodium

percentage %Na and Sodium Adsorption Ratio (SAR)

was performed. A high concentration of Na^+ may

cause the dispersion of soil mineral particles and the

decrease of water penetration (Hassen et al., 2016). The calculation of %Na ranged between 26.10 and 58.52% showing that Feriana-Skhirat groundwater is excellent to permissible for irrigation except two samples. The Wilcox (1955) diagram presenting sodium percentage and total concentration plotted in fig 5 shows that the majority of the groundwater samples are within the field excellent to good except two samples that fall in the field of doubtful to unsuitable for irrigation. As for Sodium Adsorption Ratio (SAR), it may be defined as a measure of alkalinity and sodium risk to plants. Excessive sodium concentration relative to low calcium and magnesium contents may induce to saturation of the cation exchange complex which can destroy the soil structure and reduces the soil permeability (Kumar et al. 2007). To ameliorate a water classification for irrigation suitability, the SAR has been plotted with the salinity measurement on USSL diagram (Richards, 1954) in figure 4. In Feriana-Skhirat aquifer, the SAR value ranged from 1.27 to 6.83 and as seen in the figure 4, the majority of the samples fall in C3-S1 filed except two samples that fall in C4-S2. The C3-S1 filed indicates high salinity and low sodium water. This type of water can irrigate different types of soil with little hazard of exchangeable sodium (Subramani et al. 2005) The two samples that fall in the field of C4-S2 indicate very high salinity and medium alkalinity hazard.Thus, this water can be used for plants that tolerate high salt concentration Drinking water: The chemical analyses of groundwater and the comparison with the WHO (2008) guidelines or drinking water are represented inTab 1. It indicates that the average concentration of TDS, major ions are lower than the permissible limit recommended by the WHO. Table 1 also presents the weights assigned to the different physicochemical parameters depending on its relative importance in the global quality of water for drinking purposes.The highest weight (5) were assigned toTDS and and NO_3^- owing to their major significance in water quality assessment and their health implications when they have high concentration in water (Serinivasamoorthy et al., 2008; Yiadana et al., 2010a).The minimum weight of 1 has been assigned to the parameter HCO_3^- and SO_4^{2-} due to the least importance in water quality assessment (Varol and Davaraz, 2014).The computed WQI values ranges from 21.3 to 58.6. According to the WQI type of water, 100% of groundwater samples of Feriana-Skhirat aquifer defined “Excellent Water” 5 CONCLUSIONS Feriana Skhirat aquifer, located in Central Tunisia, has been explored by geochemical methods in order to identify the mean factors and processes controlling groundwater chemistry. The concentrations of major ions are within the permissible limit for drinking purposes. The sequence of abundance of cations and

Figure 4. Salinity and Alkalinity hazard of Irrigation water

in US salinity diagram.

Figure 5. Suitability of groundwater for irrigation in Wilcox

Diagram.

anions follow the following order: SO_4 > HCO_3 > Cl and Na > Ca > Mg > K. However, the Piper classification of Feriana-Skhirat aquifer distinguished two groups: SO_4^{2}-Ca^{2+}-Mg^{2+} facies and SO_4^{2-}-Na^{+} facies.

The hydrochemical data interpretation reveals that the mineralization is mainly regulated by water/rock interaction through mineral dissolution and probably by ions exchange processes enhanced by long groundwater residence time.

As for groundwater assessment, irrigation indices (SAR, %Na) indicate that most of the groundwater samples of Feriana-Skhirat aquifer were suitable for

Hamzaoui-Azaza F., Ameur M., Bouhlila R., Gueddari M .2012. Geohemical Characterization of groundwater in a Miocene Aquifer, Southeastern Tunisia. Environ Eng Geocsci 18:159–174.

Hamzaoui F., Tlili B., Bouhlila R., Gueddari M. 2013a. Suitability of groundwater of Zeuss-Koutine aquifer (Southern of Tunisia) for domestic and agricultural use in Water Quality. In: Human Impact and Environmental Health ISBN978-1-62417-111-6. NOVA, USA, pp. 109–130.

Hamzaoui-Azaza F., Tlili-Zrelli B., Bouhlila R., Gueddari M.2013b. An integrated statistical methods and modeling minerals-water interaction to identifying hydrochemical processes in groundwater in Southern

Tunisia. CHEM SPEC BIOAVAILAB 25(3):165-178.
doi:10.3184/095422913X138569075430
Hassen I. 2013. Modélisation hydrogéologique des nappes de Kasserine. Master thesisi, University of Tunis el Manar
Hassen I., Bouhlila R., Hamzaoui-Azaza F., Khanfir R. 2014 Hydrogeological Modeling of Kasserine auifer sys tem, Central Tunisie. 10th International Hydrogeological Congress of Greece/Thessaloniki 223-230.
Hassen I., Hamzaoui-Azaza, F. and Bouhlila R. 2016. Appli cation of Multivariate statistical Analysis, Hydrochemical and Isotopic Investigations for evaluation of groundwa ter quality and its suitability for drinking and agriculture purposes: Case of Oum Ali Thelepte Aquifer, Central Tunisia. Environmental Monitoring and Assessment DOI: 10.1007/s10661-016-5124-7.
Khanfir R. 1980. contribution à l'étude hydrogéologique de la région d'Oum Ali Thelepte (Kasserine).phD Thesis, University of Pierre and Marie Curie.
Krishna Kumar S., Rammohan V., Dajkumar Sahayam J. and Jeevanandam M. 2009. Assessment of groundwa ter quality and hydrogeochemistry of Manimuktha River basin, Tamil Nadu, India. Environmental Monitoring and Assessment. 159 (1-4): 341-351.
Kumar M. 2004. An integrated hydrochemical and isotopic study of NCR-Delhi, India [in English]. Mphil. Jawaharal

Nehru University.

Kumar M., Kumari K., Ramanathan A.l., Saxena R. 2007.

A comparative evaluation of groundwater suitability

for irrigation and drinking purposes in two inten

sively cultivated districts of Punjab, India. Environ Geol

53:553–574.

Kumar M., Jyoti, P. D. Al Ramanathan , B. 2015. Integrated

hydrogeochemical, isotopic and geomorphological depic

tion of the groundwater salinization in the aquifer system

of Delhi, India. Journal of Asian Earth Sciences. doi:

10.1016/j.jseaes Li, P., Wu, J., Qian, H. 2012. Groundwater Quality Assessment Based on Rough Sets Attribute Reduction and TOPSIS Method in a Semi-arid Area, China. Environmental Monitoring and Assessment 184(8): 4841–4854. Li, P., Wu, J., Qian, H. 2013. Assessment of groundwater quality for irrigation purposes and identification of hydrogeochemical evolution mechanisms in Pengyang County, China. Environmental Earth Science 69: 2211–2225. Meybeck, M. 1987. Global chemical weathering of surfıcial rocks estimated from river dissolved loads. American Journal of Science 287: 401–428. Ouda B. 2000. Paleohydrologie isotopique du bassin de Maknassy. (Tunisie Centrale) pendant le quaternaire récent, phD Thesis, University of Sfax, Tunisia Parkhurst D.L., Appelo C.A.J. 1999. User’s guide to PHREEQC (version2): a computer program for speciation, batch reaction, one dimensional transport, and inverse geochemical calculations. US Geol.Surv.Water Resour.Inverst.Rep Richards L.A. 1954. Diagnosis and improvement of saline alkali soils.US Department of Agriculture Ramesh, K., and Elango, L.2011. Groundwater quality and its suitability for domestic and agricultural use in Tondiar river basin, Tamil Nadu, India. Environmental Monitoring Assessment doi: 10.1007/s10661-011-2231-3 Subramani T., Elango L. and Damodarasamy S.R. 2005. Groundwater quality and its suitability for drinking and agricultural use in Chithar River Basin,Tamil Nadu, India. Environmental Geology 47:1099–1110. Varol, S., and Davraz,A. 2014. Evaluation of the groundwater quality with WQI (Water Quality Index) and multivariate analysis: a case study of theTefenni plain

(Burdur/Turkey). Environmental Earth Science doi: 10.1007/s12665-0143531-z Wu, J., Li, P., Qian, H., Duan, Z., Zhang, X. 2014. Using correlation and multivariate statistical analysis to identify hydrogeochemical processes affecting the major ion chemistry of waters: a case study in Laoheba phosphorite mine in Sichuan, China. Arabian Journal of Geosciences 7: 3973–3982. World Health Organization (WHO) .2008. Guidelines for drinking water quality. vol. 1 recommendations (3rd). Geneva: WHO Wilcox L.V .1955. Classification and use of irrigation water. In Washington, DC, USA Yangui H., Zouari K., Rozanski K .2012. Hydrochemical and isotopic study of groundwater in Wadi El HechimGaraa Hamra basin, CentralTunisia. Environ Earth Sci 66: 1359–1370. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Modelling the transport and decay of microbial tracers in a macro-tidal estuary

A.A. Bakar, R. Ahmadian & R.A. Falconer

School of Engineering, Cardiff University, Cardiff, UK

ABSTRACT

The Loughor Estuary is a macro-tidal coastal basin, which is located along the Bristol Channel, in the South West of the U.K. This estuarine region can experience severe coastal flooding during high spring tides, as a result of the maximum spring tidal range of 7.5 m, occurring near Burry Port.As the Loughor Estuary and the surrounding coastal waters provide natural sites for recreational bathing and shellfish beds, with water quality in this water basin being important for com pliance with the designated standards, as to secure the safety of the bathing and shellfish harvesting industries. The estuarine system, however, potentially

gets impaired by receiving bacterial overloads from catchments through combined sewer overflows and rivers (Kay et al. 2008), besides the extreme inter tidal flooding at Llanrhidian saltmarsh, and from the lower industrial and residential areas at Llanelli and Gowerton.

A two-dimensional hydro-environmental model has been developed to simulate the hydrodynamic and tur bulence processes in the Severn Estuary and Bristol Channel. The model has been refined and extended to include the highest water level, covering the inter tidal floodplains of the estuary, and with integration of LiDAR data with the bathymetry of the model.

Four different microbial tracers – Serratia marcescens, Enterobacter cloacae, MS2, and φX174 phages, were isolated from the seawater and sewage and could be inactivated by extremes of pH, heat and solar radia tion (Wyer et al. 2014), are released at the Loughor Estuary from four different locations.

The transport of these tracers were calibrated for advection and diffusion, by considering the bed fea tures of the inter-tidal floodplains and dispersion due to turbulence and flushing during slack tides by river discharges. The decay of the released microbial tracers were modelled as the simple first-order rate functions, with the time required to exponentially inactivate 90%

Building water and chemicals budgets over a complex hydrographic network

V. Carbonnel

Department of Water Pollution Control, CP208, Université Libre de Bruxelles (ULB), Belgium

N. Brion, M. Elskens & P. Claeys

Laboratory of Analytical, Environmental and Geo-chemistry, Vrije Universiteit Brussel, Belgium

M.A. Verbanck

Department of Water Pollution Control, CP208, Université Libre de Bruxelles (ULB), Belgium

ABSTRACT

The Brussels Metropolitan Community (BMC) is a densely populated, trans-regional area of circa 800 km 2 (about 2,000,000 inhabitants), over which Brussels economy and urbanization will thrive and expand in the coming decades. The interconnected system composed of the Zenne and the Charleroi-Brussels-Scheldt ship canal (Fig. 1), which supports the complex hydrographic network in the BMC, con stitutes the backbone for the sustainable development of the economy in this area.

Profoundly modified over the last two centuries to address several issues such as flooding and water pollution, this hydrographic network constitutes a paradigm of a complex system submitted to multiple types of human perturbations

impacting the hydrologic functioning, the water courses,

the water budget, and also, directly or indirectly, the water

quality. In addition, as it spans over the three Belgian Regions

Figure 1. Scheme of the water transit lines in the BMC. Red

circles represent the limits of the box model (see text). Table 1. List of the followed parameters and tracers. Group Parameter Source Basic Conductivity, pH, O 2 , suspended matter Tracers K, B Sewage Nitrogen NH 4 Sewage/in stream NO 3 Agriculture/in stream Metals Pb, Zn, Cd, Cu, Ni, Hg Industry/urban runoff Isotopic δ 11 B Sewage δ 15 N-NH 4 Sewage/in stream δ 15 N-NO 3 , δ 18 O-NO 3 Agriculture/in stream δ 206 Pb, δ 207 Pb, δ 208 Pb Industry/urban runoff δ 66 Zn Industry/urban runoff Pesticides Diuron, isoproturon Agriculture PAH Fluoranthene, Urban runoff benzo(k)fluoranthene, Urban runoff benzo(b)fluoranthene Urban runoff (Flanders, Brussels and Wallonia, Fig. 1), the management of this system is divided in sections which are under the supervision of different organisms and administrations. Our general objective was to study the propagation of the anthropogenic disturbances in this aquatic system that goes through such a densely populated and economically active area, with multiple connections and sources of pollutants. We thus established a box-model representation of water and polluting chemicals budgets for the interconnected Zenne and Canal on the whole BMC area, with the inner nodes set at the regional boundaries (red circles in Fig. 1). As identifying and quantifying every individual chemical is a tremendous and costly task, this aim was achieved by combining hydrological and sediment dynamics analyses with the use of selected traditional and innovative chemical tracers representing different types of human activity: agriculture, industries and urban surface runoff and sewage (Table 1). Data were retrieved from: 1) several large existing datasets compiled in each region, 2) hydrological and suspended particle transport modeling, and 3) field campaigns conducted during different seasons at box model boundaries and, for background conditions, at the sources of water streams representative for the catchment. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Evaluating the ecological restoration of a Mediterranean reservoir

P. Sidiropoulos & M. Chamoglou

Management Body of Ecodevelopment Area of Karla – Mavrovouni – Kefalovriso – Velestino, Stefanovikio, Greece

I. Kagkalou

Democritus University of Thrace, School of Engineering, Department of Civil Engineering, Sector of Hydraulics,

Xanthi, Greece

ABSTRACT: The restoration of Lake's Karla environment and ecosystem is studied through the evaluation of its current status. Its drainage, in 1962, created a series of environmental problems and led to the local economy contraction. The Lake Karla reconstruction project has begun since 2000. The construction of a reservoir at the lowest part of Lake Karla watershed is among the parts of this project, which will be supplied from Pinios River and the runoff of the surrounding mountains. This newly re-established water body is considered a vital aquatic ecosystem as it is listed in the network of Natura 2000 and has been characterized as a Permanent Wildlife Refuge by Greek Law. The results indicate that the delay of implementation of Lake Karla reconstruction project, the violation of Environmental Terms and the lack of environmental policy are the most important cause of pressures.

1 INTRODUCTION

Lake Karla (Thessaly, Greece) was considered one of the most important shallow lakes in Greece until 1962, where complete drying of the lake took place – creating more agricultural land – and is now being re-constructed, establishing a 'new' reservoir offering multi services (i.e social, economic and ecological sus tainable development of the region). Prior to 1960's Lake Karla was considered one of the most important hydro-ecosystems in the region as it served as a natu ral reservoir providing water storage and recharge to groundwater. Accordingly the importance of restoring this water body and reversing the environmental con ditions caused by manmade activities was considered of high importance by the European Union. The con cept of the Lake Karla' restoration project is, probably, the first European project towards nature conservation following the ecotechnological approach.The Karla's project has been recognized as an important oppor tunity for addressing the complex challenges in the hydro-ecological management in the main agricultural region of Greece with growing demand of freshwater, providing also the traditional and new services to the people.

In this paper we discuss the restoration effort of Karla Lake as well as the pressures and their causes that negatively affect this restoration. This is achieved by evaluating the results of the monitoring and fieldwork programs that Management Body of Ecodevelopment Area of Karla – Mavrovouni – Kefalovriso – Velestino performs the last four years funded by the European

Union. 2 THE STUDIED SITE-DATA DESCRIPTION 2.1 The biogeographical context Former Lake Karla occupied the lowest part of its natural basin and was considered as one of the most important wetlands in Greece until 1962 (Fig. 1). Surface runoff from the watershed and floodwaters of the Pinios River (discharging via the Asmaki ditch) supplied the lake with large quantities of freshwater. In terms of biodiversity, the former Lake Karla endowed with a variety of habitats (pelagic, floating vegetation, shallow marshes with Juncus sp. and Typha sp., emergent vegetation and rocks), had the ability to support a rich fish and bird fauna (Jerrentrup 1990). The structure and function of Lake Karla was intimately linked with the Pinios River. The river occasionally overflowed, and floodwaters rich in oxygen and nutrients drained into Karla. Because of its gently sloping bed, lake surface area fluctuated between 4000 and 18,000 ha, depending on the inflow-outflow balance. Much of the surrounding farmland was inundated when floodwaters were held in the lake, but today, the river is leveled to prevent flood damage. The climate of the area is typical continental with cold and wet winters and hot and dry summers. Mean annual precipitation in the lake's watershed is about 560 mm and is distributed unevenly in space and time. Mean annual potential evapotranspiration is about 775 mm and the mean annual temperature is 14.3 ∘ C (Vasiliades et al. 2009). The geological structure consists mainly of recent grains of various sizes originating from the lake's deposits. The plain consists of aquiferous,

essentially sandy intercalations separated by layers of clay to silty-clay and is bound by schist and karstic limestones or marbles. The decision for the complete drying of the lake in 1960s took place in order to cre ate more land for agriculture and to avoid the flooding of the low elevation lands because of its surface area fluctuations.

2.2 The project design

Technical studies, recommended draining the lake via the Karla Tunnel and building a smaller reservoir instead of the natural lake for flood protection and for the revelation of agricultural fields.The re-constructed Lake Karla occupies the lowest part of the former Lake Karla. It lies between latitude 39 ∘ 26 ′ 49 ′′ S to 39 ∘ 32 ′ 03 ′′ N and longitude 22 ∘ 46 ′ 47 ′′ W to 23 ∘ 51 ′ 50 ′′ E and has a surface of 38 km^2 . It is characterized by its shallow depth with a maximum water depth of 4.5 m and a mean depth of 2 m.

Among the targeted measures were:

- The establishment of riparian zones to control the diffuse agricultural pollution sources.
- The operation of a peripheral buffer zone for the elimination of the nutrients input.
- The establishment of habitats corridors to avoid the ecosystem fragmentation and to improve the re-habilitation.
- The re-operation of a controlled outlet to improve flushing effect minimizing the water retention time.
- The construction and operation of artificial wetland for the improving of water quality.
- The construction of fish ladders at Pinios pumping station.
- The re-organization and the modification of the agricultural strategy in the whole region according to agri-environmental friendly practices.

– Finally, the restoration plan included the designa tion of the area as a protected Eco development area under the Natura 2000 network because of its bio diversity and the establishment of a Management Body which is responsible for the implementation of the restoration plan, the monitoring program, and the application of management principles taking into consideration the local socio-economic needs.

2.3 The post-monitoring program

A monitoring program was deployed just after the establishment of the Management Body focusing on the water quality along with the trophic conditions, the hydrological budget and the new wetland's biodi versity. The European Directives (i.e the Water Frame work Directive and the Habitat Directive) were used as guidelines for the applied methodology, for the charac terization of the water body and for the assessment of its status. More details about the sampling and moni toring methodology concerning nutrients (Dissolved Inorganic Nitrogen, DIN, Soluble Reactive Phos phorus, SRP), turbidity (Secchi depth), Chlorophyll

a as a proxy of algal biomass, and cyanotoxicity Figure 1. Aerial photograph of Lake Karla in 1945 (after Ananiadis 1956). regarding the concentration of the total amount of Microcystins have been described in detail and published (Sidiropoulos et al. 2012, Chamoglou et al. 2014). Fish were sampled in early April in three successive years (2011–2013) using multimesh benthic gillnets, type Nordic

(1.5 m × 30 m) of 12 mesh-sizes (5, 6.25, 8, 10, 12.5, 15.5, 19.5, 24, 29, 35, 43 and 55 mm, knot-to-knot) according to the CEN standard (CEN 2005). The sampling effort was provided by the CEN standard, and depended on the area of the lake (38 km^2) and its maximum depth (<2.4 m) (CEN 2005). Nets were set in late afternoon (18:00–20:00) and retrieved the following morning (6:00–8:00) in order to ensure a constant fishing effort. Complementary, for the fish species that could not be captured with nets, electrofishing was conducted in spring around the lake according to the CEN standard (CEN 2003). All fishes caught were identified according to Kottelat and Freyhof (2007) and measured for total length (TL, cm ± 0.1) and total weight (W, g ± 0.1). In the present paper the hydrological regime, the eutrophication trends along with biodiversity issues are discussed under the light of the pressures and the undertaken measures. 3 RESULTS The design of the restoration project and the establishment of the protected area have been addressed the increase of the provisional services as well as the promotion of regulating services and yet, the reestablishment of the cultural services delivered in the wide area named: Eco development Area of KarlaKefalovrysou-Mavrovouniou-Velestino (Fig. 2). In the former Lake Karla, there was no a natural surface outlet of the aquatic system Pinios River-Lake Karla, while it is thought that discharge took place

Figure 2. Map of Eastern Thessaly showing the boundaries of Lake Karla Watershed and Ecodevelopment Area. The monitoring stations (fish fauna and water quality) are also depicted.

through a system of karstic sinkholes of Mavronouni Mountain to Pagasitikos Gulf (Fig. 2). Nowadays, this discharge has been avoided with the creation of a non flow embankment, where the new reservoir meets the kartsic structures of Mavrovouni Mountain. So, the new Lake Karla is supposed to be drained artifi cially, by means of a tunnel (Fig. 2) to the Pagasitikos Gulf, which – at present – is closed and operates only in emergency situations. Thus, water losses from Lake Karla are through evaporation, withdrawals for irrigation and recharge to the aquifer.

Figure 3 demonstrates the water level fluctuation in lake Karla during the years 2012, 2013 and 2014. The lowest ecological limit as it has been set up by the Environmental Terms of the project should be at +46.4 m in order to support the functions of the wet land. It comes clear that the water level in the new reservoir was always lower than this cut off, except once, on the 11/06/2012. This happens because the reservoir receives less water inputs since 2012 both from its basin because of unfinished works, and from Pinios River due to an interruption in their connection but also due to the withdraws before the river inflows the reservoir. During the winter period, precipitation feeds the lake while, at the moment, the Karla reser voir experiences intraannual water level fluctuations due to the bad and unsustainable management of the inflows from Pinios river.

3.1 Water quality/eutrophication

Extended monitoring of water quality took place throughout the years 2011, 2012, 2013 in three sam pling stations while the sampling sites design was modified for the fish fauna sampling (Fig. 2). In table 1 the descriptive values of the key-eutrophication Figure 3. Water lever fluctuation of Lake Karla for the years 2012, 2013 and 2014. Table 1. Average concentration of key-eutrophication parameters of Lake Karla water during 2011–2013. Parameter /Units Mean Min Max DIN (mg/lt) 0.35 0.001 1.737 SRP (mg/lt) 0.063 0.004 0.461 Chla (mg/cm 3) 136.65 5.12 1136.03 Secchi depth (m) 0.34 0.19 0.49 MCYSTs (μg/lt) 1.423 0.28 4.61 Figure 4. Percentage (%) participation of fish family caught in Lake Karla in terms of a) abundance and b) biomass. parameters and of the dominant type of cyanotoxins (i.e total Microcystins expressed as MCYSTs) are presented. Concerning the biodiversity issues in the new lake Karla, at the moment we are focusing on fish fauna since endemic fish species are protected by European and national legislation. During our study the fish community of the new reservoir is composed by five families and nine species with the family of Cyprinidae being the most dominant both in terms of abundance (75,83%) and biomass (76,34%) (Fig. 4). In table 2

Table 2. Assessment of fish fauna in the protected area referred to in Article 4 of Directive 2009/147/EC and listed in Annex

II of Directive 92/43/EEC and site evaluation for them. Species Population in the site Site assessment

A/A Code Scientific Name Type Abundance Category Conservation Isolation Global

1 5256 Barbus sperchiensis p P B A A

2 1103 Alosa fallax r V C A A

3 5309 Cobitis vardarensis p P C C A

4 5307 Cobitis stephanidisi p P C C A

Code: The four character sequential code for each species can be found in the reference portal

Type : P=permanent, r=reproducing, c=concentration, w=wintering

Abundance categories : C=common, R=rare, V=very rare, P=present

Conservation : A=excellent, B=good, C=average or reduced

Isolation:A=almost isolated, B=not-isolated, but on margins of area of distribution, C: population not-isolated within extended

distribution range

Global: A=excellent value, B=good value, C=significant value

the conservation status of the protected fish fauna in the area is presented according the European Directive (92/43/EE) for the period 2011–2013.

4 DISCUSSION

Lake Karla, in terms of typology (WFD) is considered as an artificial lake, and furthermore, as a highly modi fied water body. Since there are not established pristine reference conditions for this type based on biological data, as it is suggested by the Water Framework Direc tive EU 2000/60/EC (WFD), the classification into categories is mainly based on expert judgment. Based on the present data, and taking as reference conditions for Greece, the oligotrophic reservoir of Tavropos lake (Katsiapi et al. 2012) and based on our experience we could argue that the water status of lake Karla falls into 'poor' or 'bad' category.

The environmental conditions of the new reservoir have not improved in spite of the undertaken measures. The new reservoir is characterized as hypereutrophic with the frequent occurrence of algal blooms domi nated by cyanobacteria (Chamoglou et al. 2014). Lake

Karla receives all types of pollution acting as a sink for pollutants, suspended matter and toxics. Eutroph ication produces a shift in the biological structure of reservoirs and lakes. A growing phytoplankton com munity feeds on the increased amounts of available nutrients and produces a turbid environment, which affects all life forms. In Karla reservoir there has been recorded an increase to algal biomass along with a shift in the zooplankton population towards small-size species (Stabouli et al. 2012).

At present, Lake Karla has an extremely high water retention time since there is no outflow (Chamoglou et al. 2014) thus shaping the phytoplankton commu nity and promoting cyanotoxins producing species (Papadimitriou et al. 2013). The observed cyanotox icity, which also favored by the eutrophic, warm character of the reservoir and by the hydrological alter ations posing a serious threat for the public health (WHO 1998). With increasing warming (Loukas et al. 2014) it may therefore be more difficult to fulfill the suggested ecological state targets of the lake, without undertaking additional efforts to reduce nutrient loading to levels lower than the present-day expectations. During the studied period in reservoir of Karla fish community composed by nine species and five families with the family of Cyprinidae being the most abundant which is also characteristic for the shallow warm lakes of Southern Europe (Beklioglou et al. 2007). Although the only available study before Lake Karla’s drying up indicated signals of eutrophication (Ananiadis 1956), we suggest that the re-constructed Lake Karla is experiencing a severe eutrophication. Lake Karla’s refilling process includes the supply of large quantities of Pinios River water, which is characterized by high nutrient loads (Bellos & Savvidis 2005) thus Pinios River has a large influence on the lake’s quality since its basin drains and receives waste from the most agricultural area in the country. 5 CONCLUSION To our opinion, the “alarming” current status of Lake Karla ecosystem is a synergy of three factors: 1. Decline of Environmental Terms of the project (Ministry of Environment, Planning and Public Works 2000) and failures in the planning of the project. According to the Environmental Terms the scheduled wetland, (in the west part of the reservoir,) ought to be constructed firstly thus filtering the inflows. Yet, the water level of reservoir has never been over the lowest ecological stage (+46.4 m.) that the Environmental Terms commands. The artificial wetland which will upgrade the water quality has not been operated yet thus there is a nutrient rich inflow into reservoir. 2. The great delay of the Lake Karla project implementation leading to a hysteresis of the ecological restoration of the site. The only work that has

been completed is the construction of the reser

voir, the pumping stations and the water transfer

project from the Pinios river. The collectors, who

will supply the reservoir with flushing water from

the surrounding mountainous area, have not been

constructed yet. Irrigation work, which will act

as an “outflow” thus enhancing flushing, is still

uncompleted.

3. Lack of environmental policy due to fragmentation

of responsibilities, and further due to competent

authorities dealing with water management where

in some cases their responsibilities overlap. More

over, due to the fact that the active involvement of

the local communities is very weak. Increased par ticipation in the whole basin management decisions by a wide range of stakeholders has been widely advocated by the Management Body of Lake Karla Regarding at biodiversity issues the new Karla's reservoir could be characterized as a 'hot-spot" whereas as a Natura protected area could lie at the very core of Europe's Green Infrastructure. It could not only act as an important reservoir for biodiversity and healthy ecosystem, but also could deliver many ecosystem services to society.The re-constructed Lake Karla will success to regain its multiple functional role only if we respect the characteristics of the former Lake Karla in relation to present environmental and socioeconomic needs.

ACKNOWLEDGEMENTS

This research has been co-financed by the Euro pean Union (ERDF: European Regional Development FundERDF) and Greek Nation in the framework of the Operational Program "Environment and Sus tainable Development" 2007-2013 – Priority Axis "Protecting Nature and Biodiversity".

Ananiadis, C.I. 1956. Limnological study of lake Karla. Bull.

Inst. Oceanologr. 1083: 1-19.

Beklioglu, M., Romo, S., Kagalou, I., Quintana, X., Becares,

E. 2007. State of the art in the functioning of shallow

Mediterranean lakes: Workshop conclusions. Hydrobiolo

gia 584: 317-326.

Bellos, D. & Savvidis, T. 2005. Chemical pollution moni

toring of River Pinios (Thessalia, Greece). J. of Environ.

Manage. 76(4): 282-292.

CEN, 2003. Water quality – Sampling of fish with electricity.

CEN/TC 230, Ref. No. EN 14011,16 pp.

CEN, 2005. Water quality – Sampling of fish with multimesh

gill nets. Ref. No 14757. Brussels, 27pp. Chamoglou, M., Papadimitriou, Th., Kagalou, I. 2014. KeysDescriptors for the Functioning of a Mediterranean Reservoir: The Case of a New Lake Karla-Greece. Environ. process. 1: 127-135 European Commission, 1992. Directive 92/43/EEC, Conservation of natural habitats and of wild fauna and flora. Official Journal of the European Communities No L 206/7/1992. European Commission, 2000. Directive 2000/60/EC of the European Parliament and the Council of the 23 October 2000 establishing a framework for Community action in the field of water policy. Official Journal of the European Communities. L 327:1-73. Jerrentrup, H. 1990. The fauna of the former Lake Karla. In: P. Gerakis (ed.), Conservation and management of the Greek wetlands: 527-535. Thessaloniki: Greece. Joint Ministerial Decision, 2000. Re-flooding of former Lake Karla at Perfectures of Larissa and Magnisia, Athens: Greece (in Greek). Katsiapi, M., Michaloudi, E., Moustaka-Gouni, M., Pahissa Lõpez, J. 2012. First ecological evaluation of the ancient Balkan Lake Megali Prespa based on plankton. J. Biol. Res. 17: 51-56. Kottelat, M., Freyhof, J. 2007. Handbook of European freshwater fish. Kottelat,Cornol and Freyhof, Berlin Loukas, A., Kagalou, I., Laspidou, C.H., Vasiliades, L., Papadimitriou TH., Sidiropoulos, P. 2014. Evaluation of restored lake Karla status and operation under climate change scenarios. 10th International hydrogeological congress of Greece, 8-10 October, Thessaloniki, Greece. (in Greek). Ministry of Environment, Planning and Public Works, 2000. Joint Ministerial Decision 112839/18-12-2000: Modification – Supplementation – Codification of the Environmental Terms for the construction and operation of former lake Karla re-flooding project at Prefectures of

Larisa and Magnesia, Athens: Greece (In Greek). Papadimitriou, Th., Katsiapi, M., Kormas, K., MoustakaGouni, M. Kagalou I. 2013. Artificially-born “killer” lake: phytoplankton based water quality and microcystin affected fish in a reconstructed lake. Sci. Total Environ. 452–453: 116–124. Sidiropoulos, P., Papadimitriou, Th., Stabouli, Z., Loukas,A., Mylopoulos, N., Kagalou, I. 2012. Past, present and future concepts for conservation of the re-constructed lake Karla (ThessalyGreece). Fresen. Environ. Bull. 21(10a.2012): 3027–3034. Stabouli, Z., Papadimitriou, Th., Kagalou, I. 2012.Assessment of the ecological status of Lake Karla, using populations of zooplankton. Congress of E.E.B.E. Trikala: Greece. Vasiliades, L., Loukas, A., Patsonas, G. 2009. Evaluation of a statistical downscaling procedure for the estimation of climate change impacts on droughts. Nat. Hazard. Earth SyS. 9: 879–894. World Health Organization, 1998. Cyanobacterial toxins: microcystin-LR. In Guidelines for drinking water quality. 2nd Edition, Addendum to vol.2. Health criteria and other supporting information: 95–110. Geneva: Switzerland. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Spatio-temporal interaction of morphological variability with hydrodynamic

parameters in the scope of integrative measures at the river Danube

M. Glas, M. Tritthart, M. Liedermann, P. Gmeiner & H. Habersack

Christian Doppler Laboratory for Advanced Methods in River Monitoring, Modelling and Engineering; Institute of

Water Management, Hydrology and Hydraulic Engineering; Department of Water, Atmosphere and Environment;

University of Natural Resources and Life Sciences, Vienna, Austria

ABSTRACT: Hydrodynamic parameters, determined by 3D numerical models, provide a basis for simulation

tools such as sediment transport and habitat models. Whereas most of the time measures are evaluated using

models right before and after measure implementation, little is known about the interaction of hydrodynamics, sediment transport and river morphologies including their spatio-temporal variability. This study investigates that interaction in the scope of a river engineering project at the Danube, containing integrative measures. Different morphologies led to a significant difference in hydrodynamics (e.g. bed shear stress: +11.8% at specific locations regarding its mean values). Morphodynamic models coupled with 3D hydrodynamics allow – in contrast to 1D/2D models – simulation times of a few months only within reasonable computational time. Hence, for the purpose of assessing measures, scenarios with high and low morphologies, embodied as an initial condition, should be considered for morphodynamic models or habitat models to cover effects of spatio-temporal variability.

1 INTRODUCTION

Assessing measures in rivers, e.g. river restoration and navigation aspects, is widely done by using numerical tools to gain a priori or post-measure knowledge of the interaction of hydrodynamics, sediment transport and river morphologies as well as habitat quality. The quality of the assessment depends on both the qual ity of the applied numerical code and the quality of the investigated data and scenarios. By implementing morphodynamic models on temporal scales of several decades, a restriction to relatively simple numerical codes is inherited due to their more feasible com putation times. On the other hand, sophisticated 3D numerical models (e.g. Olsen, 1999; Fluent, 2003; Tritthart, 2005) restrict simulation time to durations far below those time scales due to their numerical demands (Tritthart et al., 2011).

When applying such sophisticated codes measures in rivers are often assessed by numerical models being applied only right before and after the measure, due to the high demand in data, skills and time. Thus, little effort is made in investigating the variability within morphologic, hydrodynamic and sediment transport processes before and after the measure, when highly sophisticated 3D numerical codes in combination with a data basis highly variable in space and time provide the foundation for a detailed analysis of this variabil ity. A set of integrative measures was applied in a

river reach in the Austrian Danube (Habersack et. al., 2012) suitable for this analysis due to the high amount and quality of data available, acquired from several measurement campaigns. The aim of this study was (i) to evaluate variability in bed elevation and hydrodynamic parameters, determined from bathymetry and 3D hydrodynamic model, respectively; (ii) to draw conclusions of this variability on sediment transport models and ecological aspects; and (iii) finally to figure out consequences for planning and assessing future measures. 2 STUDY REACH The study was conducted at a 3 kilometer long river reach within the free flowing section of the Austrian Danube east of Vienna – near the city of Hainburg – between river kilometers 1887,5 and 1884,5 (Fig. 1). It is part of the national park ‘Donau Auen’ and represents a transboundary waterway. Thus, the river faces several ecological and economical demands. Channel degradation and negative sediment budget in the past have resulted in river bed incision and habitat loss. 2.1 Integrative measures Several integrative measures were applied between autumn 2012 and autumn 2014. An alternative groyne layout – characterized by an attracting and partially convoluted scheme with an increased groyne distance,

Figure 1. Study reach including an overview of the implemented integrative measures. River kilometers are marked with a

cross.

reduced groyne heads and lowered groyne roots – was implemented and a method called granulometric bed improvement (Habersack et al., 2012; Lieder mann et al., 2014) was tested to cope with ongoing river bed incision (Habersack et al., 2008). Side arm reconnection for low flow conditions was performed in combination with river bank restoration applied mainly at the right shore, including the removal of groynes and rip rap. The integrative measures were accompanied by a detailed monitoring concept, includ ing hydrodynamics (water tables, flow velocities), sediment transport (bed load, suspended load, bed layer composition), ground water as well as ecolog ical aspects (e.g. fish, invertebrates, vegetation and birds). Numerical models serve as a key factor to provide spatio-temporally distributed data for all mon itoring aspects. Hence, knowledge on the variability of morphology and hydrodynamics is crucial.

2.2 Hydrology

The hydrology of the upper Danube catchment is char acterized by highly variable discharge. Mean annual discharge MQ is 1930 $m^3 s^{-1}$; low annual discharge (94% exceedance duration) is represented by a value of 980 $m^3 s^{-1}$, normally occurring in autumn or winter. Floods are appearing in spring and summer; a flood with a return period of 100 years is characterized by 10,400 $m^3 s^{-1}$ peak flow. Within the observed period of 10 years an extreme flood event occurred in June 2013 (>100 year return period, 10,500 $m^3 s^{-1}$ peak flow). Several flood events with a peak flow of around 7300 $m^3 s^{-1}$ (10 year return period) occurred as well. Figure 2 shows the hydrograph and the analyzed model geometries within the observed period, subdivided into five model geometries prior to the measures, one during measure implementation, and four thereafter. Each model is characterized by discharge values observed over the period of 3 months prior to the date of the corresponding bed level survey, indicated by boxplot diagrams. Obviously, models from 2006/04 and 2013/07 exhibit higher and more variable discharge values due to preceding extreme flood events (10 year and >100 year return period, respectively). Hence, flood events were likely to yield an impact on the morphology applied within the models.

3 MODELS

3.1 Digital elevation model

Ten digital elevation models (DEM) were created by merging Airborne Light Detection and Ranging (LIDAR) data with up-to-date multi-beam riverbed bathymetry over the course of ten years (Figure 2). River structures were partially enhanced with geodetical measurements (theodolite, DGPS). As the multibeam river bed bathymetry represents the dataset with the highest variability over time, DEM were matched to the recording time of the multi-beam measurements.

3.2 3D numerical model

The 3D numerical model RSim-3D (Tritthart 2005), specifically developed for rivers, was applied successfully within several studies concerning hydrodynamics, sediment transport, and habitat modelling (e.g. Tritthart & Gutknecht, 2007; Hauer et al., 2011; Schludermann et al., 2012, Lechner et al., 2014).The numerical solution procedure, based on a Finite Volume Approach, approximates the RANS equations based on an unstructured polyhedral computation mesh. Turbulence closure is provided by the k-ε turbulence scheme. Within this study, 3D numerical models for mean flow conditions were set up for each of the ten

Figure 2. Hydrograph of the study site indicated as solid grey line. Box plot diagrams indicate discharge values of a 3 month

period ending at the time when hydrodynamic models – created from updated DEM – were determined. Grey solid line within

the boxplots represents mean values. Grey dotted line represents discharge at mean flow conditions.

evaluated DEM. Calibration and validation of the

models was performed using several measured water

tables, ADCP and ADV measurements. Details for

models prior to the measures are given in Tritthart et

al. (2009). Mean annual discharge MQ (1930 ms^{-1})

was set as a constant inflow condition. Outflow condi

tions, implemented as a fixed water level, were derived

from the current rating curve at the gauge Hainburg

(river kilometer 1883.92), provided by the Austrian

waterway authority ‘viadonau’. Calibration yielded

an absolute roughness value of 0.03 m for instream

regions, and a value of 0.30 m for river structures and

floodplains.

4 RESULTS AND DISCUSSION

4.1 Modelled bed shear stress

Modelled shear stress derived from the ten 3D numeri

cal models was analyzed in relation to its morphology.

In Figure 3 river bed elevation and the correspond

ing modelled bed shear stress are shown. Apparently,

models of flood-affected morphologies showed a dis

tinct characteristic: The occurrence of a deep scour

at river kilometer 1887.0 corresponded to low bed

shear stress values, while a relatively high river bed

between river kilometer 1886.2 and 1885.4 led to high

shear stress values of up to 21 Nm -2 . Thus, flood

affected morphologies were likely to shift towards a

river bed, representative for mean flow conditions in

between extreme flood events. Furthermore a substan

tial difference between preand post-measure models

was discernible in both river bed elevation and mod

elled bed shear stress – especially within the section

between river kilometer 1887.0 and 1886.0 – indicat

ing the change of morphodynamic and hydrodynamic

processes induced by the integrative measures. Finally,

a considerable variation within pre-measure models

(not flood-affected) was observed.

Figure 4a and Figure 4b indicate the spatial distribu

tion of bed shear stress between preand post-measure Figure 3. Longitudinal section along the river axis for (a) bed elevation and (b) bed shear stress. Grey lines indicate pre-measure states; black lines represent states during or after the measure was initiated. Dotted lines show flood-affected states. models, respectively. Low reduction values of bed shear stress (2007/12, Fig 4a) were ranging from 0 to 3 Nm^{-2} within the main channel, whereas groyne fields were characterized by an increase of bed shear stress values up to 6 Nm^{-2}. High reduction values of bed shear stress (2008/03, Fig 4b) of 1 to 5 Nm^{-2} were visible within the main channel from river kilometer 1887.5 to 1885.75, followed by a small area of increased values. Groyne fields showed increased values as well for this geometry. Compared to that, the flood-affected model geometry during measure implementation revealed a different spatial distribution. Increased values were discernible between river kilometer 1887.0 and 1886.25

Figure 4. Spatial distribution of the difference of bed shear stress (Nm^{-2}) between the post-measure model 2014/09 and the

pre-measure models (a) 2007/12, (b) 2008/03 and (c) 2013/07. Grey shaded areas represent a reduction. River kilometers are

marked with a cross.

(0 to 3 Nm^{-2}). The upstream part of the groyne fields

showed increased values too, whereas the downstream

part exhibited reduced values. The section between

river kilometer 1886.25 and 1885.0 was characterized by an increase of bed shear stress up to 6 Nm^{-2}.Again, the retreat from a flood-affected morphology to a natural one is likely to occur during mean flow conditions according to high values at opposite locations.

Figure 5. Variability of modelled hydrodynamic parameters

over time (2007/03 to 2012/09) for the range of (a) water depth

H; (b) depth averaged flow velocity U; and (c) bed shear stress

τ along a longitudinal section (river axis)

4.2 Variability of hydrodynamic parameters

Due to the short periods of time between post-measure models, variability within modelled hydrodynamic parameters was evaluated for pre-measure models only. Moreover, post-measure models were influenced by dredging operations intended to sustain a sufficient fairway depth for navigation.

Figure 5 presents the range of hydrodynamic parameters (water depth H, depth-averaged flow velocity U and bed shear stress τ) for pre-measure models (mean flow) except flood affected ones. A difference of variability along the river axis was discernible. The maximum range of water depth ($H_{max} - H_{min}$) of 0.81 m (10.3% deviation from mean water depth) occurred at the deep scour at river kilometer 1886.85. Downstream of the scour – between river kilometer 1886.60 and 1886.20 – variability of depth-averaged flow velocity and bed shear stress increased as a consequence of variable water depth at the scour and the resulting variability in water table gradient. The mean range of depth-averaged flow velocity along the river axis ($U_{max} - U_{min}$) was 0.09 ± 0.04 ms^{-1}. A maximum range of 0.17 ms^{-1} (+8.8% deviation from mean depth-averaged flow velocity) was found at river kilometer 1886.15.

The mean range of modelled bed shear stress along the river axis ($\tau_{max} - \tau_{min}$) was 1.69 ± 0.62 Nm^{-2}. A maximum range of 3.05 Nm^{-2} (+11.8% deviation

from mean bed shear stress) was located at river kilo

meter 1886.20. As a consequence, the variability of

bed shear stress was found to play an important role

in the assessment of measures with respect to the

long term development of sediment transport (river morphology) and should be investigated for other discharges as well. Furthermore, the variability of the analyzed parameter should be investigated for effects on assessing habitat quality. 4.3 Implications on sediment transport modelling and ecological aspects Morphodynamic models coupled with 3D hydrodynamics have the potential to yield improved results over 1D and 2D models due to their improved hydrodynamic representation of the flow field. At the same time unsteady simulation times of only a few months are feasible within reasonable computation time. However, shorter simulation times increase the influence of the starting condition, especially the DEM, representing morphology at the beginning of the simulation. Even higher influences of the starting condition (DEM) are to be considered for simulations of steady flow conditions, as hydrodynamics are changing only slightly at distinct locations. To tackle that issue, the authors suggest to investigate the sensitivity of the starting condition by running at least two scenarios for hydrodynamics and sediment transport models. One should inherit low bed levels and the other one high bed levels of the initial DEM, to cover extreme scenarios for the assessment of planned measures concerning sediment transport. In the case of the study presented, models based on the DEM from 2007/12 and 2008/03 could cover those extreme scenarios. Additionally, habitat models or other ecological analyses dealing with the analyzed hydrodynamic parameters could profit from this sensitivity study. 4.4 Consequences for planning and assessing future measures When considering future measures a special focus should be given to a substantial preand postmonitoring, including (i) a sufficient number of bathymetry data sets to gain extreme scenarios from real data and (ii) a morphodynamic and numerical analysis of those extreme scenarios. 5 CONCLUSIONS The variability of hydrodynamic parameters over time was evaluated using a 3D-numerical model for mean flow conditions within an integrative river project. For different initial bed morphologies, maximum ranges of modelled hydrodynamic parameters were 0.81 m, 0.17 ms^{-1}

and 3.05 Nm^{-2} for modelled water depth, depth-averaged flow velocity and bed shear stress, respectively. Concerning bed shear stress, the implemented river engineering measures led to a reduction of up to 3 Nm^{-2} for a scenario with relatively low bed levels. In contrast, the reduction for a scenario with high bed levels amounted to 1 to 5 Nm^{-2} . This difference potentially has an effect on sediment transport models, when considering short simulation times.

Therefore a sensitivity study of the used morphology should be conducted when assessing future measures. This is of even higher importance when only short term simulations are possible – i.e. due to the use of complex numerical codes – and initial conditions are becoming more relevant for the outcome. Moreover, estimated variabilities of hydrodynamics could be relevant also for other assessment studies, e.g. of habitat quality.

ACKNOWLEDGEMENT

The financial support by the Austrian Federal Ministry of Economy, Family and Youth and the National Foundation for Research, Technology and Development is gratefully acknowledged. The study presented partly contributed to the monitoring task of the Integrated River Engineering Project on the Danube East of Vienna, co-financed by ‘viadonau’, the European Union and the Austrian Ministry of Transport, Innovation and Technology.

Fluent Inc. 2003. Fluent 6.0 User’s Guide. Lebanon. New Hampshire, USA. 2003

Habersack, H., Hour, C., Liedermann, M. & Tritthart, M.

2008. Modelling and monitoring aid management of the

Austrian Danube. Water 21, Dec. 2008, 29–31.

Habersack, H., Liedermann, M., Tritthart, M., Hauer, C.,

Klösch, M., Klasz, G. & Hengl, M. 2012. Maßnahmen für

einen modernen Flussbau betreffend Sohlstabilisierung

und Flussrückbau—Granulometrische Sohlverbesserung,

Buhnenoptimierung, Uferrückbau und Gewässervernet

zung. Österreichische Wasserund Abfallwirtschaft 64:

571–581. Hauer, C., Unfer, G., Tritthart, M., Formann, E. & Habersack, H. 2011. Variability of mesohabitat characteristics in riffle-pool reaches: Testing an integrative evaluation concept (FGC) for MEM-application. River Research and Applications 27(4): 403–430. Lechner, A., Keckeis, H., Schludermann, E., Loisl, F., Humphries, P., Glas, M., Tritthart, M., & Habersack, H., 2014. Shoreline configurations affect dispersal patterns of fish larvae in a large river. ICES Journal of Marine Science 71(4): 930–942. Liedermann, M., Gmeiner, P., Tritthart, M., Glas, M. & Habersack, H. 2014. Occurrence and characterization of bed forms at the Danube to the east of Vienna. In: CRC Press/Balkema, Proceedings of the International Conference on Fluvial Hydraulics, RIVER FLOW 2014, 1075–1080. Olsen N.R.B. 1999. Computational Fluid Dynamics in Hydraulic and Sedimentation Engineering, Class Notes, Norwegian Univeristy of Science and Technology, Trondheim, Norway. Schludermann, E., Tritthart, M., Humphries, P. & Keckeis, H. 2012. Dispersal and retention of larval fish in a potential nursery habitat of a large temperate river: an experimental study. Canadian Journal of Fisheries and Aquatic Sciences 69(8): 1302–1315. Tritthart, M. 2005. Three-Dimensional Numerical Modelling of Turbulent River Flow using Polyhedral Finite Volumes. Vienna: Wiener Mitteilungen Wasser-AbwasserGewässer Tritthart, M. & Gutknecht, D. 2007. 3-D computation of flood processes in sharp river bends. P I Civil Eng-Wat M 160: 233–247 Tritthart, M., Liedermann, M. & Habersack, H. 2009. Modelling spatio-temporal flow characteristics in groyne fields. River Research and Applications 25(1): 62–81. Tritthart, M., Liedermann, M., Schober, B. & Habersack, H. 2011. Non-uniformity and layering in sediment transport modelling 2: river application. Journal of Hydraulic Research 49(3): 335–344. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Analysis of contributions and uncertainties of fish population models for the development of river continuity concepts in the river basin Ruhr, Germany

D. Teschlade & A. Niemann

Institute of Hydraulic Engineering and Water Resources Management, University of Duisburg-Essen,

Essen, Germany

ABSTRACT: The European Union and Germany aims to increase the share of renewables. Thus, new and sus

tainable hydropower solutions are discussed in detail. Simultaneously, the European Water Framework Directive

(WFD) prohibits the degradation of rivers, which leads to the need of river continuity concepts and new fish

friendly hydropower concepts that do not (significantly) impact connectivity and continuity. Migratory obstacles,

particularly in the form of hydropower plants, have a significant impact on aquatic ecosystems. In order to ensure

efficient energy production that does not interfere with river ecology, but provides a reasonable contribution to the

power supply, fish and water protection measures are indispensable. However, there is a need for comprehensive

research, in the areas of fish protection and fish decline, particularly in the fields of behavioral and population

biology of both diadromous and potamodromous species. These knowledge gaps have significant consequences

for the acceptance of measures for fish protection, fish migration and consequently as well for the extension or

upgrading of hydropower. This paper presents an innovative assessment approach by development of a model that

links physical habitat conditions and ecological requirements of species communities and different life stages

using a habitat modelling software (MaxEnt) and a dispersal modelling software (FIDIMO) to derive priorities

for protected reaches. As a first step we demonstrate that reliable spatial predictions for key fish species can be

achieved by modelling habitat suitability with species occurrence data and public available hydromorphological

variables.

1 INTRODUCTION

Rivers belong to the most stressed and intensively

human-influenced ecosystems in the world. Numer

ous anthropogenic obstructions like water diversions

for using hydropower and other industrial purposes,

irrigation and domestic uses are the reason why most

of the world's largest rivers are nowadays fragmented

by dams (Jager et al., 2001).

The functional connectivity of river catchments

plays an important role for both aquatic and terres

trial organisms. It allows the colonization of habitats,

genetic exchange and leads to a temporal connec

tivity of different habitats within the seasons or life

cycle stages (Fausch et al., 2002). Fragmentations are

associated with range reductions (Perkin et al., 2013)

which are particular important for fish as fish migrate depending on age and seasons to different special habitats (Ebel, 2013). These inhibited (long distance) migrations have a severe impact on the biocenosis and leads to the decline of number of individuals and shifting in biocenosis To counter these and other adverse effects the European Water Framework Directive (WFD) (Directive No 2000/60/EC) came into force in 2000 (EC, 2000). The WFD was launched by the EU with the basic goal of ensuring that all types of water bodies attain a “good ecological status”. The ecological status is primarily classified in terms of biological quality elements, complemented by abiotic and hydrogeomorphological elements that include river connectivity (Mader and Maier, 2008). The restoration of the continuity of watercourses for “aquatic organisms and sediments” is according to Irmer (2000) an essential condition to reach a “good ecological status” of surface waters. To achieve the WFD, in consideration of a large number of migratory obstacles, river continuity concepts are needed. For this purpose the identification of the most critical barriers for fish migration is necessary, which is one of the main aims of this project. By combining species habitat models (MaxEnt) with species dispersal models (FIDIMO) we are able to show the effects of habitat suitability, dispersal, and fragmentation on the distribution of river fishes. These information can be used to develop the methodological basis for the analysis of ecological continuity needs. 1.1 River restoration challenges In order to achieve the goals of the WFD, numerous river restoration measures have been implemented in recent years. Restoring fish migration can have a substantial positive effect on fish distribution, productivity and abundance (Fullerton et al., 2010, Roni et al., 2002) for which reason the removal of all barriers should be the preferred solution (Brevé et al., 2014).

However, such measures can’t be implemented in any anthropogenically influenced regions like Europe or especially Germany, due to various functions of water bodies such as the use of hydropower. Other technical solutions are needed to solve the problem of ecological continuity in our rivers.

Multiple reference books to choose and design proper fishpasses are available. In Germany a certain state of the art for the design of fishpasses (upstream migration) was developed (DWA, 2014). Fishpasses are customized for target fish species considering geo metric and hydraulic limited values (e.g. discharge, flow velocity, energy dissipation, pool dimension etc.). As construction method both natural and technical fishpasses are included.

The functioning and operation of fish passes to ensure the upstream migration is well studied, but still there is a lack of knowledge concerning downstream migration and fish protection at hydropower plants as well as ethology and population biology of diadromous and potamodromous species (Naumann and Heimerl, 2013). Hence a lot of research is carried out currently. New findings usually originate from ethohydraulic experiments, which are validated using field investiga tions. Simultaneously, several numerical models have been developed, which have the big advantages that boundary conditions can be changed easily to analyze different impacts (Rutschmann et al., 2014). However, there is still a need for comprehensive research, in the areas of fish protection and fish decline, particularly in the fields of behavioral and population biology of both diadromous and potamodromous species. These knowledge gaps have significant consequences for the acceptance of measures for fish protection and fish migration.

In consideration that not all dams may be removed there is an urgent need to develop a procedure to pri oritize most critical barriers. In this case one of the main current challenges in restoration fish migration is according to Teichert et al. (2016) the use of large databases of biological monitoring surveys (e.g.WFD) to help environmental managers prioritizing restora tion measures to address the connectivity problem more strategically (Williams et al., 2012). It has to clarify whether we are able to describe sufficiently robust responses in order to use them as guidelines for the restoration of river connectivity. In this regard physical habitat models (Mouton et al., 2007) and statistical models to predict the likely occurrence or distribution of species based on relevant variables (Ahmadi-Nedushan et al., 2006) became an important tool for river conservation planning and manage ment. Like for other ecological processes, modeling

can improve our understanding by generalizing and

simplifying complex systems to a small set of key

components (Breckling et al., 2011).

Referring to this multiple approaches for quantify

ing habitat availability have recently been developed

(see reviews: Fullerton et al. (2010) & Kemp and

O'Hanley (2010)). Moreover, several studies to pri

oritizing barriers for remediation based on barrier Figure 1. Ruhr catchment area, Germany (Ruhrverband, 2013). permeability and maximizing fragment sizes have already been carried out. (e.g. (Brevé et al., 2014, Meixler et al., 2009, Cote et al., 2009)). A deficit of these models is that assessment criteria are often based on swimming capabilities of upstream migrating adult salmonids, while ignoring other life-stages, nonsalmonid species, downstream migration and behavior (Kemp and O'Hanley, 2010). 2 MATERIAL AND METHODS 2.1 Study area The study area is located in western Germany, in the southeast of North-Rhine-Westphalia (Fig. 1) and comprises the whole catchment (4.478 km 2) of the River Ruhr (51 ∘ 12 ′ 48 ′′ – 51 ∘ 27 ′ 3 ′′ N, 8 ∘ 33 ′ 28 ′′ – 6 ∘ 43 ′ 23 ′′ E).The river is about 219 km long as a tributary of the River Rhine. In the upper course the catchment area is characterized by agricultural and forestry landuse. The western part belongs to the Ruhr area, which is one of the biggest European industrial area with about 5 million inhabitants. The western part is determined by urban and industrial areas. The catchment area was selected as the study area for the previously stated issues. Due to industrialization, population growth as well as rising utilization pressure in the past, there is a significant anthropogenic pressure in the shape of the Ruhr and the Ruhr catchment area itself. Apart from technical measures, the ecosystem of the river was also affected by building dams. Nowadays the waters in the Ruhr catchment area are important for many different reasons such as drinking water supply and sewage disposal. Extensive anthropogenic impact on the environment in general, but also past intense technical river improvements are the reason that most of the rivers in the catchment (69%) are “Clearly Disturbed” (class 4) or in worse condition. In the River Ruhr catchment area currently exist approx. 1.300 transverse structures (>20 cm threshold height) of different sizes whereof many of the bigger ones used for producing hydropower. According to Pottgiesser and Sommerhäuser (2008) the River Ruhr and its tributaries can be assigned to the Table 1. Stream types for all relevant rivers in the River Ruhr catchment area. LAWA Total Type Description Percent length 5 Small coarse substrate dominated siliceous highland rivers 67,6% 1.251,5 km 5.1 Small fine substrate dominated siliceous highland rivers 0,7% 13,0 km 6 Small fine substrate dominated calcareous highland rivers 1,4% 25,1 km 7 Small coarse substrate dominated calcareous highland rivers 6,7% 39,7 km 9 Mid-sized fine to coarse substrate dominated siliceous highland rivers 12,4% 187,4 km 9.1 Mid-sized fine to coarse substrate dominated calcareous highland rivers 0,5% 8,9 km 9.2 Large highland river 10,7% 124,0 km

following short names of the biocoenotically relevant

stream types for Germany:

2.2 Data

In Europe, the WFD requires the assessment of hydro

morphological quality in establishing the ecological

status of rivers from all EU Member States. In Ger

many habitat variables are obtained from a state-wide

hydromorphological survey that assessed the hydro

morphological status of the rivers according to Ger

many's Working Group of the Federal States on Water

Problems Issues (LAWA). For the assessment, the

streams are divided into segments (50–100 m sections)

serving as survey units.The proven basic concept oper

ates as follows: the local scale habitat variables are

grouped into 31 single parameters, which are then

aggregated into six main parameters ((1) River course,

(2) longitudinal profile, (3) riverbed structure, (4) cross section profile, (5) riparian structure and (6) river environment) (Gellert et al., 2014). For stream prop erties, we calculated stream order (Shreve/Strahler), gradient and distance from mouth using both hydrog raphy data set feature and digital elevation model for stream segments with ArcGIS Hydrology tools.

2.3 Fish sampling

We were able to acquire observed occurrence data of different fish species for the study area from the State Agency for Nature, Environment and Consumer Pro tection North Rhine-Westphalia and the Department of Aquatic Ecology (Faculty of Biology) of the University of Duisburg-Essen.

A total of 578 sites in the river basin Ruhr were sampled between 2002 and 2012.The main data source for model calibration and validation was performed using electrofishing, which is the least biased method for sampling fish. In total 42 species in 14 families were detected (Table 2).

2.4 Modelling approach

This project provides a modeling framework to develop the methodological basis for the analysis of ecological continuity needs by using the method of Radinger and Wolter (2015). They combined species habitat models

(MaxEnt; (Elith et al., 2011)) with species dispersal

models (FIDIMO; (Radinger et al., 2014)) to show the Table 2. Number of fish species (>10 individuals) at 578 sampling sites. Family and common name Scientific name Pres. Anguillidae European eel Anguilla anguilla 187 Cyprinidae Ide Leuciscus idus 31 Barbus Barbus barbus 55 Common bream Abramis brama 26 European chub Squalius cephalus 154 Eurasian minnow Phoxinus phoxinus 155 Gudgeon Gobio gobio 94 Eurasian dace Leuciscus leuciscus 46 common carp Cyprinus carpio 23 Roach Rutilus rutilus 85 Tench Tinca tinca 29 Bleak Alburnus alburnus 17 Salmonidae Grayling Thymallus thymallus 102 Atlantic Salmon Salmo salar 24 Brown trout Salmo trutta fario 413 Rainbow trout Oncorhynchus mykiss 117 Petromyzontidae River Lamprey Lampetra fluviatilis 26 Europ. Brook Lamprey Lampetra planeri 83 Percidae European perch Perca fluviatilis 91 Eurasian ruffe Gymnocephalus cernua 49 Gasterosteidae Three-spined stickleback Gasterosteus aculeatus 136 Cottidae Bullhead Cottus gobio 351 Esocidae Northern pike Esox lucius 51 Nemacheilidae Stone loach Barbatula barbatula 204 Siluridae Wels catfish Siluru glanis 12 effects of habitat suitability, dispersal, and fragmentation on the distribution of river fishes (Radinger and Wolter, 2015). One of the main goals is to ensure a transferability to other catchments by using existing environmental variables only. 2.5 Habitat suitability Habitat models, or species distribution models (SDM; also referred to ecological niche models or habitat

models), are important tools to explore the effects

of changes on biodiversity at different spatial scales

(Jones et al., 2012, Guisan and Thuiller, 2005). They

are using a set of habitat components to predict some

attributes of wildlife populations and serve three main

purposes: (1) to predict species occurrences on the

basis of abiotic and biotic variables, (2) to improve the

understanding of species-habitat relationships and (3)

to quantify habitat requirements (Ahmadi-Nedushan

et al., 2006). In recent years there have been many developments in the field of species distribution mod eling, and multiple methods are now available.According to Elith et al. (2011) a major distinction among methods is the kind of species data they use. Habitat suitability models can be generated using methods requiring information on species presence or species presence and absence.

For modeling the habitat suitability of the Ruhr catchment area (4.478 km 2) the maximum entropy model MaxEnt (Phillips et al., 2006) is used due to the fact that the analyses will based on presence data only. MaxEnt is a machine-learning approach to predictive niche modeling that quantifies the relation between the occurrences of a species (presence only) (Elith et al., 2011, Franklin, 2010) MaxEnt uses a generative approach to estimate the currently known environmental variates conditioning species presence and bases the final prediction on the principle of maximum entropy. This specifies that the best approximation of an unknown distribution is the probability distribution with maximum entropy, subject to the constraints imposed by the sample of species presence observations (Phillips et al., 2006).

2.6 Species dispersal

The dispersal of animals is a key process in community ecology. Populations are often subdivided in space, spread among habitats of highly variable quality that are connected via larval transport and adult migration (Yakubu and Fogarty, 2006). The dispersal governs the exchange between subpopulations, the emigration and immigration as well as the (re)-colonization of new habitats and strongly influences the spatio-temporal distribution and abundance of species (Radinger et al., 2014). Due to this, dispersal models became an important role in recent decades. An overview of the previously used fish dispersal models can be found in (Radinger et al., 2014). Radinger et al. (2014) showed that s significant disadvantage of all these models was that fish dispersal along river networks has never been implemented in geographic information systems.

Radinger et al. (2014) developed an open-source GRASS GIS model to assess species-specific fish dis persal within the catchment. The fish dispersal model, called “FIDIMO”, calculates species spread as a lep tokurtic diffusion process from predefined sources (presence points) described by a double-normal prob ability density function to account for stationary and mobile dispersal-relevant components of a fish population (Radinger et al., 2014). Figure 2. ROC curves for the MaxEnt model prediction averaged on 10 replicate runs for River Ruhr target fish species. The average test AUC is 0.850 for thymallus thymallus (grayling). The dispersal kernels were derived from a literature review of refitted dispersal kernels describing the movement of 62 fish species in 160 data sets (Radinger and Wolter, 2014). A dispersal kernel determines how the fish units move within the river network (Muneepeerakul et al., 2008). Based on these refitted dispersal kernels, provided a multiple regression model that allows predicting the species-specific mean movement distances of the stationary and the mobile component from four factors: (1) fish length, (2) aspect ratio of the caudal fin, (3) stream size (stream order), and (4) time. For each species, the model calculates dispersal as a diffusion process from all source points to all other sites in the catchment (Radinger et al., 2014). 3 FIRST RESULTS As a first step we used the maximum entropy method (MaxEnt) to model habitat suitability with the abovementioned species occurrence data and environmental variables. In total we considered 44 hydromorphological environmental layers and the most frequently target species thymallus thymallus (grayling) for the River Ruhr catchment area. 3.1 Model performance evaluation Model performance was evaluated with receiver operating characteristic (ROC) analysis, which has been widely useful for SDM’s with threshold independent outputs (Phillips et al., 2006). A typical ROC curve plots true positive rate (sensitivity) against falsepositive rate for the entire range of possible thresholds, therefore providing a unified representation for assessing the overall model performance (Liang et al., 2013). The area under the curve (AUC) ranges from 0 to 1, where a score of 1 indicates perfect discrimination and a score of 0.5 implies predictive discrimination no better than a random guess (Phillips et al., 2006).

Our results indicate that the model developed here

can be used to predict the likely distributions of

the investigated species. According to Hosmer et al.

(2013) interpretation, the developed target fish species

model for thymallus thymallus provided “excellent”

(>0.80) predictions. The ROC curve for the average

model prediction with a mean test AUC value (0.850

for thymallus thymallus (Grayling)) and corresponding

standard deviation (0.020) are presented in Figure 2.

3.2 Variable contributions and threshold effects

The results of the MaxEnt simulation show that the sin

gle most influential predictor for the presence/absence

Figure 3. Response curves for the analyses relating

thymallus thymallus (grayling) occurrence to the top four

most influential geographical and hydromorphological pre

dictors.The curves show the mean response of the 10 replicate

MaxEnt runs (red) and the mean +/one standard deviation

(blue, two shades for categorical variables). Positive fitted

function values suggest that species respond favorably and

low values suggest the opposite.

Figure 4. Predicted habitat suitability for thymallus thymallus (grayling) in the River basin Ruhr based on 80 sampling sites

and 44 hydromorphological variables of the investigated species was the riverbed width (54.9 %). A threshold effect was evident at approximately eight meter riverbed width for thymallus thymallus (grayling). In addition to that also special structures in form of wood, latitudinal variance and flow diversity are predictors for the species (Fig. 3). 3.3 Species habitat suitability maps A predicted grayling distribution map (Fig. 4) prepared with MaxEnt and ArcGIS for showed habitat suitability for the River Ruhr catchment. Areas with higher predicted habitat suitability mostly matched the distribution of recently observed (2002–2012) grayling occurrences. In areas where the species has not been reported, the predicted habitat suitability appeared to be constantly low (yellow). Most occurrence locations were placed in areas with a relatively high habitat suitability (green). Highly suitable habitats were predicted within gaps between existing occurrence locations. This is due to the fact, that the hydromorphological boundary conditions are nearly the same like for observed grayling occurrences. 4 CONCLUSION AND OUTLOOK During the first research about the feasibility of this project and first modelling results it becomes clear, that population models are in general suitable for the development and derivation of prioritized river continuity concepts. The MaxEnt model for the River Ruhr catchment indicates that it can be used to predict the likely distributions of the investigated species. The available environmental variables provide good conditions and can be used for further analysis. First results of habitat modeling show deficits and key habitats for respective species which are important background information for decision making.

After creation of habitat suitability maps for each fish species we will start to analyze fish dispersal. Combining both habitat suitability and species dispersal maps we will be able to evaluate barriers in terms of the achievement of habitats.

Within the project the fish dispersal model FIDIMO will be optimized in the field of ecological continuity. Particularly a more precise description of passability of technical fishpasses and the downstream migration of fish species should be implemented in the model. For this, the existing investments are analyzed and evaluated. To improve the method we will start to evaluate various migration obstacles systematically to see their cumulative effect on fish populations. A special focus is placed on technical fish passes and downstream fish protection devices at hydropower plants and influencing factors (e.g. predators) to analyze the dynamics of fish populations. One of the main goals is to use existing data only to ensure transferability to different counties or river basins authorities and to aid the implementation of the WFD. For this purpose, various scenarios of river continuity (1. actual state of river continuity, 2. scheduled handling with migra tion barriers pursuant to the implementation roadmap under the terms of the WFD, 3. solution variants) will be simulated.

Finally we will present a concept for prioritization of migration barriers and give recommendations for the restoration of the ecological continuity.

ACKNOWLEDGEMENTS

We would like to thank the State Agency for Nature, Environment and Consumer Protection (LANUV) North Rhine-Westphalia and the Ruhrverband for providing fish and GIS data on hydromorphological features. We are also grateful to the members of the department of Aquatic Ecology of the University of Duisburg-Essen and Johannes Radinger from Leibniz institute for freshwater ecology and inland fisheries in Berlin for supporting us.

Ahmadi-Nedushan, B., St-Hilaire,A., Berube, M., Robichaud, E., Thiemonge, N. & Bobee, B. 2006. A review of sta tistical methods for the evaluation of aquatic habitat suitability for instream flow assessment. River Research and Applications, 22, 503–523.

Breckling, B., Jopp, F. & Reuter, H. 2011. Backgrounds and scope of ecological modelling: Between intellectual adventure and scientific Routine. Modelling Complex Ecological Dynamics: An Introduction into Ecological Modelling for Students, Teachers & Scientists.

Brevé, N. W. P., Buijse, A. D., Kroes, M. J., Wanningen, H. & Vriese, F. T. 2014. Supporting decision-making for

improving longitudinal connectivity for diadromous and potamodromous fishes in complex catchments. Science of The Total Environment, 496, 206–218. Cote, D., Kehler, D., Bourne, C. & Wiersma, Y. 2009. A new measure of longitudinal connectivity for stream networks. Landscape Ecology, 24, 101–113. DWA 2014. Fischaufstiegsanlagen und fischpassierbare Bauwerke: Gestaltung, Bemessung, Qualitätssicherung ; Merkblatt DWA-M 509. Ebel, G. 2013. Fischschutz und Fischabstieg an Wasserkraftanlagen. Handbuch Rechen-und Bypasssysteme. Bd, 4. EC 2000. European Parliament and the Council of the European Union. Directive 2000/60/EC establishing a framework for the Community action in the field of water policy. Official Journal of the European Communities, L327, 1–72. Elith, J., Phillips, S. J., Hastie, T., Dudík, M., Chee, Y. E. & Yates, C. J. 2011. A statistical explanation of MaxEnt for ecologists. Diversity and Distributions, 17, 43–57. Fausch, K. D., Torgersen, C. E., Baxter, C. V. & Li, H. W. 2002. Landscapes to riverscapes: bridging the gap between research and conservation of stream fishes a continuous view of the river is needed to understand how processes interacting among scales set the context for stream fishes and their habitat. BioScience, 52, 483–498. Franklin, J. 2010. Mapping species distributions: spatial inference and prediction, Cambridge University Press. Fullerton, A. H., Burnett, K. M., Steel, E. A., Flitcroft, R. L., Pess, G. R., Feist, B. E., Torgersen, C. E., Miller, D. J. & Sanderson, B. L. 2010. Hydrological connectivity for riverine fish: measurement challenges and research opportunities. Freshwater Biology, 55, 2215–2237. Gellert, G., Pottgiesser,T. & Euler,T. 2014.Assessment of the structural quality of streams in Germany—basic description and current status. Environmental Monitoring and Assessment, 186, 3365-3378. Guisan, A. & Thuiller, W. 2005. Predicting species distribution: offering more than simple habitat models. Ecology Letters, 8, 993-1009. Hosmer, J. D. W., Lemeshow, S. & Sturdivant, R. X. 2013. Model-Building Strategies and Methods for Logistic Regression. Applied Logistic Regression. John Wiley & Sons, Inc. Irmer, U. 2000. The new EC Framework Water Directive: Assessment of the chemical and ecological status of surface waters. Acta Hydrochimica Et Hydrobiologica, 28, 7-14. Jager, H., Chandler, J., Lepla, K. & Van Winkle, W. 2001. A Theoretical Study of River Fragmentation by Dams and its Effects on White Sturgeon Populations. Environmental Biology of Fishes, 60, 347-361. Jones, M. C., Dye, S. R., Pinnegar, J. K., Warren, R. & Cheung, W. W. L. 2012. Modelling commercial fish distributions: Prediction and assessment using different approaches. Ecological Modelling, 225, 133-145. Kemp, P. S. & O'hanley, J. R. 2010. Procedures for evaluating and prioritising the removal of fish passage barriers: a synthesis. Fisheries Management and Ecology, 17, 297-322. Liang, L., Fei, S., Ripy, J. B., Blandford, B. L. & Grossardt, T. 2013. Stream Habitat Modelling for Conserving a Threatened Headwater Fish in the Upper Cumberland River, Kentucky. River Research and Applications, 29, 1207-1214. MADER, H. & MAIER, C. 2008. A method for prioritizing the reestablishment of river continuity in Austrian rivers. Hydrobiologia, 609, 277-288.

Meixler, M. S., Bain, M. B. & Todd Walter, M. 2009. Predict

ing barrier passage and habitat suitability for migratory

fish species. Ecological Modelling, 220, 2782-2791.

Mouton, A. M., Schneider, M., Depestele, J., Goethals, P. L.

& De Pauw, N. 2007. Fish habitat modelling as a tool for

river management. Ecological Engineering, 29, 305-315.

Muneepeerakul, R., Bertuzzo, E., Lynch, H. J., Fagan, W.

F., Rinaldo, A. & Rodriguez-Iturbe, I. 2008. Neutral

metacommunity models predict fish diversity patterns in

Mississippi-Missouri basin. Nature, 453, 220-222.

Naumann, S. & Heimerl, S. 2013. Fischschutz und Fischab stieg im Dialog-Forum “Fischschutz und Fischabstieg”.

WasserWirtschaft, 103, 30–35.

Perkin, J. S., Gido, K. B., Al-Ta’ani, O. & Scoglio, C. 2013.

Simulating fish dispersal in stream networks fragmented

by multiple road crossings. Ecological Modelling, 257,

44–56.

Phillips, S. J., Anderson, R. P. & Schapire, R. E. 2006.

Maximum entropy modeling of species geographic dis

tributions. Ecological Modelling, 190, 231–259.

Pottgiesser, T. & Sommerhäuser, M. 2008. Beschreibung

und Bewertung der deutschen Fließgewässertypen—

Steckbriefe und Anhang.

Radinger, J., Kail, J. & Wolter, C. 2014. FIDIMO – A free

and open source GIS based dispersal model for riverine

fish. Ecological Informatics, 24, 238–247.

Radinger, J. & Wolter, C. 2014. Patterns and predictors of

fish dispersal in rivers. Fish and Fisheries, 15, 456–473. Radinger, J. & Wolter, C. 2015. Disentangling the effects of habitat suitability, dispersal, and fragmentation on the distribution of river fishes. Ecological Applications, 25, 914–927. Roni, P., Beechie, T. J., Bilby, R. E., Leonetti, F. E., Pollock, M. M. & Pess, G. R. 2002. A review of stream restoration techniques and a hierarchical strategy for prioritizing restoration in Pacific Northwest watersheds. North American Journal of Fisheries Management, 22, 1–20. Ruhrverband 2013. Zeit im Fluss: 100 Jahre Ruhrverband, Klartext Verlag. Rutschmann, P., Bui, M.-D., Abdelaziz, S., Yao, W., Rutschmann, B. & Geiger, F. 2014. Ökologische Modellierungen-drei Fallbeispiele. Teichert, N., Borja, A., Chust, G., Uriarte, A. & Lepage, M. 2016. Restoring fish ecological quality in estuaries: Implication of interactive and cumulative effects among anthropogenic stressors. The Science of the total environment, 542, 383–93. Williams, J. G., Armstrong, G., Katopodis, C., Larinier, M. & Travade, F. 2012. Thinking like a fish: A key ingredient for development of effective fish passage facilities at river obstructions. River Research and Applications, 28, 407–417. Yakubu, A.-A. & Fogarty, M. J. 2006. Spatially discrete metapopulation models with directional dispersal. Mathematical Biosciences, 204, 68–101. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Hydrologic and hydraulic design to reduce diffuse pollution from drained peatlands

S. Mohammadighavam & B. Kløve

Water Resources and Environmental Engineering Research Group

Faculty of Technology, University of Oulu, Finland

ABSTRACT: Peatland ditching for forestry, agriculture, and peat extraction in boreal regions has a significant effect on water quality and quantity which needs sustainable and careful management. The objective of this study is to develop new methods based on more refined hydrologic information and hydraulic design that will better predict, control and improve water management solution for ditched peatlands to reduce leaching and diffuse pollution to watercourses. This study builds on field and laboratory experiments as well as hydrologic/hydraulic modelling. Peat physical properties and monitoring data on rainfall, runoff and groundwater levels were used to build hydrological model using DRAINMOD.Also COMSOL Multiphysics commercial Software was employed to optimize different parts of treatment structures. The obtained results were used to modify existing facilities and build a full scale gravity driven hydraulic mixer with 4 m width and 65 m length which is under investigation

to improve the accuracy of CFD modelling.

1 INTRODUCTION

About one third of Finland area (~ 10 million ha) is covered by peat that is a layer of organic material accu mulated from mosses, shrubs, herbs, and small trees due to incomplete decay on an annual basis under saturated conditions. Over half of Finland's peatland area has been subject to artificial drainage during decades for forestry, agriculture, and peat extraction as fuel or other uses (Natural Resources Institute Finland 2015). Peatland ditching in boreal regions has a sig nificant effect on drainage water quality and quantity (Fig. 1) and consequently on downstream environ ment which needs sustainable and careful management (Dunn & Mackay 1996, Kløve 2001, Marttila & Kløve 2008). Several methods such as constructed wetlands, peak runoff control and chemical water treatment have been introduced to limit the environmental impacts (Tammela et al. 2010, Ronkanen & Kløve 2008, Mart tila & Kløve 2009). However, past monitoring data shows significant fluctuation in purification efficiency (Kløve et al. 2012, Heiderscheidt et al. 2015).

The objective of the presented research is to develop new methods based on more refined hydrologic infor mation and hydraulic design that will better predict, control and improve water management solution for ditched peatlands to reduce leaching and diffuse pollution to watercourses.The study builds on field and laboratory experiments as well as hydrologic/hydraulic modelling. Five sites (3 forestry drained and 2 peat extraction areas) in northern Finland were selected and extensively observed (years 2011–2014). Peat physical properties and monitoring data on rainfall, runoff

Figure 1. Sediment and nutrient leaching from ditched peatland to downstream watercourses.

and groundwater levels were used to build a hydrological model using DRAINMOD 6.1 (Skaggs 1978) to predict WTD fluctuations and drain runoff for short and long-term periods.Also several three-dimensional (3D) computational fluid dynamic (CFD) turbulence models were built using COMSOL Multiphysics 5.1 (COMSOL AB 2015) commercial Software to modify and optimize different parts of sedimentation pond structures located at the drain outlets and used in water treatment. The obtained results (Saarinen et al. 2013, Mohammadighavam et al. 2015b, Mohammadighavam & Kløve 2016b) were used to modify existing facilities and build a full scale gravity driven hydraulic mixer with 4 m width and 65 m length which

Figure 2. Maps showing (left) location of Finland in northern Europe and (right) location of the study areas in the northern Finland.

is under investigation to improve the accuracy of CFD modelling (Mohammadighavam & Kløve 2016a).

2 MATERIALS AND METHODS

2.1 Study areas

Three forestry areas in the Luohuanjoki river basin (N: 64 ∘ 36 ′ 01 ′′ , E: 25 ∘ 14 ′ 50 ′′ , Fig. 1) were selected

for this study due to extensive leaching of acid sub
stances fromAS soils. Initial drainage in forestry areas
was carried out in 1970s with a systematic pattern,
open ditches with 40 m spacing and 1 m depth. Also
two active peat extraction areas (Fig. 2), Korentosuo
(N: $64^{\circ} 52' 34''$, E: $26^{\circ} 49' 28''$) and Navettarimpi (N:
$64^{\circ} 20' 37''$, E: $26^{\circ} 30' 34''$) were selected as well due to
extensive leaching of Sediment and Nutrient Loads.
Both sites are nearly flat and are drained by a conven
tional network of main and lateral open ditches with
20 m spacing and 1 m depth. Mean annual precipita
tion is around 600 mm (1911–2011 data, (Irannezhad
et al. 2014)) and mean annual temperature is 1.75 $^{\circ}$C
(1961–2011 data, (Irannezhad et al. 2015)). The study
sites have a subarctic climate and winter, the longest
season, usually begins in November and ends in late
April (Finnish Meteorological Institute 2015).

2.2 Field observation

Extensive field observation of water table depth
(WTD), drain outflow, air temperature and pressure,
and precipitation was carried out at study areas during
several years (2011–2014). For this purposes a set of
instruments and loggers were installed in each study
area (Fig. 3) that was included: one tipping bucket rain

gauge (Model 7852, Davis Instruments Corp.) with 0.2 mm resolution, two level loggers (Solinst Levelogger model 3001, Solinst Canada Ltd.) were installed in underground pipes to record water table depth fluctuation every 30 min, one V-notch weir with two level loggers (Solinst Levelogger model 3001, Solinst Canada Ltd.) at upstream and downstream to record discharge in main ditch every 5 min, and baro loggers to record air pressure continuously every 30 min.WTD

measuring have been done continually for the whole Figure 3. Tipping bucket rain gauge (left), level logger was in-stalled in underground pipe (middle), and V-notch weir with two level loggers at upstream and downstream (right). Figure 4. Laboratory measurements to identify classification and physical properties of soil (Photo by Justice Akanegbu). period of study even during winter but due to frozen conditions rainfall and discharge measuring did only during summer and autumn season. 2.3 Field and laboratory measurements In order to identify classification and physical properties of soil, several disturbed and undisturbed soil samples were collected in different places and different depths. Soil sample analyzed and tested to determine soil main specification such as texture, moisture, hydraulic conductivity, pF curve, and other essential soil properties. In addition the laboratory experiments and tests some parameters such as hydraulic conductivity measured in the field and compared with obtain result from laboratory measurements (Fig. 4). Also extensive measurements were performed on exciting canals, sediment ponds, drains, and treatment facility to collect hydraulic data that need for CFD model calibration and validation (Fig. 5). 2.4 Hydrological modelling DRAINMOD is a field-scale, process-based, and distributed model for hydrological simulation was developed specifically for shallow water table soils (Skaggs

Figure 5. Collecting data from exciting canals, sediment ponds, drains, and treatment facility (Photo by Elisangela Heiderscheidt).

Table 1. Comparison of observed and modelled WTDs for calibration and validation years at three forestry sites

	Period	Statistical Parameter	Site 1	Site 2	Site 3
Daily water table depth (WTD)	Calibration 2011 Period (cm)	MAE	3.25	5.93	6.08
		EF	0.68	0.29	0.55
	Validation 2010 Period (cm)	MAE	5.64	6.35	8.38
		EF	0.44	0.34	0.48

1978). Last version (DRAINMOD 6.1) includes freezing, thawing, and snowmelt components, and is thus capable of calculating the effect of ice formation on the hydraulic conductivity of soil and the depth and density of accumulated snow on the ground. Soil data, meteorological data and drainage network specifica tion are main data which need to make a hydrological model using DRAINMOD. In this study, DRAIN MOD performance and accuracy to simulate and predict hydrological events in cold climate was eval uated (Mohammadighavam & Kløve unpubl.). Also DRAINMOD was employed to simulate and predict WTD fluctuation in several peatland forestry area in order to decrease the leaching of acidity which is highly depended to WTD (Saarinen et al. 2013).

2.5 Hydraulic modelling

COMSOL Multiphysics is a commercial finite ele ment software package with powerful and flexible platform that allows simulation of a wide range of dif ferent types of phenomena. COMSOL has the ability to simulate flow as stationary or as time dependent, in 2D or 3D space with different types of fluids

with consideration various geometrical and physical Figure 6. The Observed and Modelled WTD in year 2011(Cali-bration period). characteristics. In this study COMSOL was employed for simulation and optimization several part of treatment facilities such as slow mixing and sedimentation pond (Karppinen et al. 2015, Mohammadighavam et al. 2015a, Mohammadighavam et al. 2015b). Also a gravity-driven hydraulic mixer (flocculator) with around-the-end barriers has been built base of primary result of this study and used as a full-scale pilot study for a series of extensive experimental investigations and measurements to assess most popular turbulence models (Mohammadighavam & Kløve unpubl.). 3 RESULTS AND DISCUSSION Impact of peatland forestry on runoff water quality in areas with sulphide-bearing sediments; how to prevent acid surges

(Saarinen et al. 2013) Hydrological models (DRAINMOD 6.1) were built for three peatland forestry area in order to investigate the leaching of acidity derived from peatland forestry and to simulate the impact of different drainage configurations. Model evaluation was performed by calculating Mean absolute error (MAE, (Janssen & Heuberger 1995)) and Nash-Sutcliffe modelling efficiency (EF, (Nash & Sutcliffe 1970)) were used to quantify the agreement between observed during two growing seasons and predicted data. The obtain results from simulation (Table 1) shows an acceptable correlation with observed water table data at most sites (2 of 3 area) during both calibration (Fig. 6) and validation period. Typically, hydraulic conductivity needed calibration and model values were one order higher than laboratory measured values. Also, runoff prediction and timing was not always well predicted which need further research. The calibrated hydrological models were useful to assess long-term drainage impacts on hydrology (simulation since 1960) as well to evaluate the effect of WTD fluctuation on the leaching of acidity derived from peatland forestry and devise potential

ways to decrease acid leaching after drain maintenance

work.

Evaluation of DRAINMOD 6.1 for hydrological

simulations of peat extraction areas in Northern Fin

land (Mohammadighavam & Kløve unpubl.)

A new DRAINMOD version including frost condi

tions was evaluated for hydrological simulation of two

Table 2. Statistical performance of DRAINMOD in pre

dicting daily water table depth (WTD) and daily and monthly

drainage at two peat extraction areas P e r i o d S t a t i s t i c a l P a r a m e t e r K o r e n t o s u o N a v e t t a r i m p i

D

a i

l y

w

a t

e r

t a

b l

e

d e

p t

h

(W

T

D

) Calibration 2012-13 MAE 5.51 7.31 Period (cm) EF 0.92
0.89 Validation 2013-14 MAE 11.29 9.09 Period (cm) EF 0.64
0.62

D

a i

l y

a n

d

m

o n

t h

l y

d r

a i

n a

g e Daily 2013 EF -9.15 -4.77 Monthly EF 0.28 0.36 Daily 2014 EF -0.75 -30.73 Monthly EF 0.30 -4.89

Figure 7. Depiction of 2D velocity fields (m/s) and streamlines for different design of inlets, Flow direction left to right. drained (1.0 m depth and 20 m spacing) peat extraction areas in Northern Finland with considerable frost. WTD was recorded continuously during two years and the data was used for model calibration and validation. The obtain results from simulation (Table 2) showed a good correlation with observation data, at least in term of WTD during both calibration and validation period but not well enough in terms of drainage water runoff. The poor prediction of runoff could be due to the impact of controlling structures along ditches, lateral seepage, and extensive variation in surface layer of peat duo to extraction activities, and annual ditch cleaning. Frost prevented continues observations during the winter season as frost free structures could not be installed. Optimization of chemical treatment basins for peat mining runoff water treatment using COMSOL flow model (Mohammadighavam et al. 2015a) Peatland ditching and peat extraction has a significant effect on runoff water quality and downstream watercourses. Simple and low maintenance water chemical treatment basin could be a significant treatment option to prevent diffuse pollution especially in remote and rural areas. The occurrence of preferential flows within the basin decrease treatment performance, therefore the influence of different inlet design (Fig. 7) on retention time of a trapezoid cross section treatment basin (top width 8 m, bottom width 6 m, 2 m depth, 110 m length, and 160 l/s discharge) was evaluated using COMSOL.

Figure 8. Velocity gradient (G-value, s-1) fields along

hydraulic flocculators of the different barrier designs tested

(O1–O6) (di-rection of flow from left to right.

Optimization of gravity-driven hydraulic floccula

tors to treat peat extraction runoff water (Moham

madighavam et al. 2015b)

Peatland drainage water, rich in humus, sediments, and nutrients requires simple chemical purification that highly depended to flocculator performance to achieve efficient particle aggregation. COMSOL was employed to optimize gravity-driven hydraulic floccu lators. Several 3D turbulence CFD model were built to evaluate different barriers' design (Fig. 8) and geom etry ratios to achieve target G-values (40–60 s^{-1}).

Comparison of experimental data and CFD pre dictions of gravity-driven hydraulic mixer:Assessment of various turbulence models (Mohammadighavam & Kløve unpubl.)

Considerable progress in the computer processing power made CFD simulators more efficient and favor able. In this part of study the performance of three of most common turbulence models which potentially suitable for 3D simulation of fluid flow evaluated using experimental datasets. For this purpose a gravity driven hydraulic mixer with around-the-end barriers (flocculator) with 60 m length, 4 m average width and 1 m depth has been built in northern Finland and employed as a full-scale pilot study for a series of Figure 9. Full-scale around-the-end gravity-driven hydraulic flocculator under investigations using the Vector 3D Acoustic Velocimeter (Nortek AS) as part of drained water treatment fa-cilities on a peat extraction field in northern Finland. extensive experimental investigations using the Vector 3D Acoustic Velocimeter (Nortek AS) (Fig. 9). Evaluation of different turbulence models was conducted by comparison average velocity (U), velocity components (u,v,w) and turbulence kinetic energy (k) predicted by the models against obtained experimental data in terms of accuracy. 4 CONCLUSIONS In this study, DRAINMOD 6.1 was employed for hydrological simulation at several conventionally drained forestry and peat extraction areas located in Northern Finland for several hydrological years. Statistical analyses showed that the model predicted WTD fluctuations in perfect agreement with observed data while there was insufficient correlation between model-predicted drainage outflow and observed data, especially in daily resolution that need further studies. Subsurface-drained fields without control structures could be useful to collect more accurate drainage data even during frozen periods. Overall, the performance of DRAINMOD 6.1 showed a satisfactory degree of accuracy in term of WTD fluctuations simulation and prediction at drained peatland area regardless of land use under extremely cold conditions in Northern Finland.The results obtained in this study also indicate the well performance of COMSOL Multiphysics®commercial finite element Software package for 3D CFD modelling, simulation and optimization of different units of treatment facilities with a wide range of flow conditions. Using COMSOL for modelling, speed up this study and allowed our research team to evaluate different geometry, hydraulic conditions, physics, and phenomena in a reasonable time and minimum cost that was too expensive and time consuming to do that in laboratory or field conditions.

DRAINMOD and COMSOL, despite of all differ

ence in usage and logic, contains many different tools,

features, settings, and interfaces to assist researchers

and designer through all the steps of workflow but

as the other modelling software package still need

experimental data for calibration and validation.

ACKNOWLEDGMENTS

Special thanks goes to Vapo Oy for funding this

study and the authors would like to thank Maa-ja

vesitekniikan tuki,TES-Tekniikan edistämissäätiö and

OLVI-säätiö for their support.

COMSOL AB. 2015. CFD module user's guide

Dunn, S. M., & Mackay, R. 1996. Modelling the hydrologi

cal impacts of open ditch drainage. Journal of Hydrology,

179(1-4): 37-66. doi: http://dx.doi.org/10.1016/0022

1694(95)02871-4

Finnish Meteorological Institute. (2015). Seasons in Finland.

Retrieved from http://en.ilmatieteenlaitos.fi/seasons-in

filand

Heiderscheidt, E., Leiviskä, T., & Kløve, B. 2015. Chemi

cal treatment response to variations in non-point pollution

water quality: Results of a factorial design experiment.

Journal of Environmental Management, 150(0): 164-172.

doi: http://dx.doi.org/10.1016/j.jenvman.2014.10.021

Irannezhad, M., Chen, D., & Kløve, B. 2015. Interannual vari

ations and trends in surface air temperature in Finland in

relation to atmospheric circulation patterns, 1961-2011.

International Journal of Climatology, 35(10): 3078-3092.

doi:10.1002/joc.4193

Irannezhad, M., Marttila, H., & Kløve, B. 2014. Long

term variations and trends in precipitation in Finland.

International Journal of Climatology, 34(10): 3139-3153.

doi:10.1002/joc.3902

Janssen, P. H. M., & Heuberger, P. S. C. 1995. Calibra

tion of process-oriented models.Ecological Modelling,

83(1-2): 55-66. doi: http://dx.doi.org/10.1016/0304

3800(95)00084-9

Karppinen, A., Postila, H., Eskelinen, R., Heiderscheidt, E., Klöve, B., Marttila, H., Mohammadighavam, S., Torabi Haghigi, A., Tuukkanen, T., Hadzic, M., Heikki nen, K., Ihme, R., & Tolkkinen, M. (2015). Turve tuotannon vesistökuormituksen muodostuminen ja sen hallintamahdollisuuksia – SulKa-hankkeen loppuraportti. (). Helsinki: Suomen ympäristökeskus.

Kløve, B. 2001. Characteristics of nitrogen and phospho rus loads in peat mining wastewater. Water Research, 35(10): 2353–2362. doi: http://dx.doi.org/10.1016/

S00431354(00)00531-5 Kløve, B., Saukkoriipi, J., Tuukkanen, T., Heiderscheidt, E., Heikkinen, K., Marttila, H., Ihme, R., Depre, L., & Karppinen, A. (2012). Turvetuotannon vesistökuormituksen ennakointi ja uudet hallintamenetelmät [pollution load of drained water from peat mines prediction and new controlling methods]. Helsinki: Suomen ympäristö. Retrieved from www.ymparisto.fi/julkaisut Marttila, H., & Kløve, B. 2009. Retention of sediment and nutrient loads with peak runoff control. Journal of Irrigation and Drainage Engineering, 135(2): 210–216. Marttila, H., & Kløve, B. 2008. Erosion and delivery of deposited peat sediment. Water Resources Research, 44(W06406) Mohammadighavam, S., Heiderscheidt, E., & Kløve, B. (2015a). Optimization of chemical treatment basins for peat mining runoff water treatment using COMSOL flow model. IC-EPSMSO – 6th International Conference on Experiments/Process/System Modelling/Simulation/ Optimization, II 324–329. Mohammadighavam, S., & Kløve, B. (a). Comparison of experimental data and CFD predictions of gravity-driven hydraulic mixer: Assessment of various turbulence models. unpubl. Mohammadighavam, S., & Kløve, B. (b). Evaluation of DRAINMOD 6.1 for hydrological simulations of peat extraction areas in northern Finland. unpubl. Mohammadighavam, S., Heiderscheidt, E., Marttila, H., & Kløve, B. 2015b. Optimization of gravity-driven hydraulic flocculators to treat peat extraction runoff water. Journal of Irrigation and Drainage

Engineering,:04015045.
doi:10.1061/(ASCE)IR.1943-4774.0000955 Nash, J. E., & Sutcliffe, J. V. 1970. River flow forecasting through conceptual models part I — A discussion of principles. Journal of Hydrology, 10(3): 282–290. doi: http://dx.doi.org/10.1016/0022-1694(70)90255-6 Natural Resources Institute Finland. (2015). Finland – the peatland capital of the world. Retrieved from http://www.metla.fi/tutkimus/suotutkimus/tausta-en.htm Ronkanen, A., & Kløve, B. 2008. Hydraulics and flow modelling of water treatment wetlands constructed on peatlands in northern Finland. Water Research, 42(14): 3826– 3836. doi: http://dx.doi.org/10.1016/j.watres.2008.05.008 Saarinen, T., Mohammadighavam, S., Marttila, H., & Kløve, B. 2013. Impact of peatland forestry on runoff water quality in areas with sulphide-bearing sediments; how to prevent acid surges. Forest Ecology and Management, 293(0): 17–28. doi: http://dx.doi.org/10.1016/j.foreco. 2012.12.029 Skaggs, R. W. (1978).A water management model for shallow water table soils. (No. 134). Raleigh, N.C.: North Carolina State University, Water Resources Research Institute. Tammela, S., Marttila, H., Dey, S., & Kløve, B. 2010. Effect and design of an underminer structure. Journal of Hydraulic Research, 48 (2): 188–196. doi:10.1080/00221681003704202
Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Flow modelling and investigation of flood scenarios on the Cavaillon River, Haiti

A. Joseph & N. Gonomy

GNR-FAMV, Université d'Etat d'Haïti, Port au Prince, Haiti

Y. Zech & S. Soares-Frazão

Institute of Mechanics, Materials and Civil Engineering, Université Catholique de Louvain,

Louvain-la-Neuve, Belgium

ABSTRACT: The vulnerability of riverine populations confronted to rivers floods and the socioeconomic

consequences of floods remain a major concern for the Haitian public decision-makers. However, the use of

modelling tools for flood prediction and mitigation purposes is not yet well established in the country. This paper

presents the first results of an ongoing study on the Cavaillon River in Southern Haiti, with the aim of constructing

a methodology to design a suitable model for the river and to run flood scenarios. First, old topographic data were

combined with new data obtained from field measurements to construct the hydraulic model. The roughness

parameter in this model was calibrated using the scarce available water level and discharge measurements. On

this basis, flood scenarios were run using an estimate of the 100-year discharge available from previous studies.

Then, an adapted procedure to construct flood maps is presented.

1 INTRODUCTION

The flood problematic is crucial in Haiti and the vul

nerability of riverine population confronted to river

floods constitutes a major concern for Haitian deci

sion makers. However, flood prevention strategies still

need to be developed and the use of modelling tools

for this purpose is not yet well established.

The lack of established good practice methods

means that in Haiti like in many Caribbean countries,

engineers and planners are often unable to predict and

mitigate floods effectively (Lumbroso et al., 2011)

This is a crucial issue since Haiti is one of the poorest

countries in the Western Hemisphere, with eighty per cent of the population living under the poverty thresh old, and most of them depending on the agricultural sector, which is very sensitive to river overflow. Flooding events that occur regularly during the cyclonic periods are disastrous for the surrounding residents but also for the river evolution itself. As an example, during the year 1986, between the 2nd and the 3rd of March, a flood occurs in the Haitian sub-region “Les Cayes” with a total of 79 deaths and 85000 affected people. Moreover, sediment trans port and bank erosion often become uncontrollable due to catastrophic deforestation of rivers basins. As an example, the forest area for the Cavaillon River basin in Southern Haiti has decreased from 380 km 2 in 1983 to 51 km 2 in 1995. Such over-exploitation of the forest areas significantly increases the dramatic consequences of flood event. Within the frame of a cooperation project funded by ARES-CCD (Académie de Recherche et d’Enseignement Supérieur – Commission de la Coopération au Développement, Belgium), the present work is part of a broader study aiming at improving the ability of Haitian institutions to better manage their rivers and prevent or at least decrease potential damages of flood events. Part of the work is devoted to the development of a methodology aiming at constructing simple and efficient hydraulic models for Haitian rivers, based on the existing knowledge and on field surveys, and using free modelling tools, considering the limited available resources in the country. This methodology is built using the case of the Cavaillon River, located in southern Haiti, in the department Les Cayes (Figure 1). Similar problems and adapted methodologies were already experienced in some countries in the Caribbean The extreme fluvial flood hazard maps for Trinidad and Tobago were based on Shuttle Radar

Topography Mission (SRTM) DTM that is freely downloadable from the Internet. This was because no other nationally consistent DTM was available (Lumbroso et al., 2011). As part of a national flood risk assessment for St Vincent 25-year return period flood maps were produced for a limited number of watercourses (DLN Consultants, 2006). One-dimensional hydraulic modelling was undertaken using the software package HEC-RAS. The hydraulic models utilized a number of

Figure 1. Localization of the studied river reach: (a) general

map of Haiti and indication of the Cavaillon river basin, (b)

Details of the Cavaillon River.

river cross-section surveys with the floodplain topog

raphy based on a coarse DTM produced from 10-ft

(3.05 m) contour interval maps (DLN Consultants,

2006),

The method that was used to produce the maps is

flexible and based on freely available software that

means when more detailed topographic data become

available, the maps can be updated quickly (Lumbroso

et al., 2011).

The paper is organized as follows. First the study

site and the available data are presented. Then, the

hydraulic model of the river is described, with the cho

sen upstream and boundary conditions. This model is

calibrated using the available discharge and water level

data, and used to study flood scenarios on the river.

Finally, the results are discussed and perspectives are given for future work.

2 DESCRIPTION OF THE STUDY SITE

The Cavaillon River flows over approximately 50 km

(Figure 1) until reaching the sea. The studied area has

a length of 25 km, located between the Dory weir and

Grand Place.

Before the start of the project, the only topo

graphical data available for the region consisted in

a 30 m × 30 m digital elevation model (DTM) which

is not sufficient considering that at many locations,

the width of minor bed of the river is less than

30m. Besides, no bathymetrical data were available.

Following a field survey (Joseph et al., 2015), 96 Figure 2. Bed profile of the Cavaillon River. Figure 3. Cross-sections in the Cavaillon River (a) upstream end of the study reach (Dory, x = 0 km) and (b) downstream end of the study reach (Grand-Place, x = 23.016 km). Figure 4. Bed material size distribution at Dory (x = 0 km, blue curve) and Grand Place (x = 23.016 km, red curve). cross-sections were measured along this reach, yielding the bed profile illustrated in Figure 2. Typical cross sections are shown in Figure 3. For each cross section, the Pebble Count method (Wolman, 1954) was applied to obtain the grain size distribution. Results for the two cross-sections of Figure 3 are shown in Figure 4: it can be observed that

Figure 5. Evolution of the median diameter d 50 along the

study reach.

Table 1. Measured discharges and water depths. PK13.115 PK18.815 PK20.984 PK23.016

	PK13.115	PK18.815	PK20.984	PK23.016
Q (m^3/s)	h (m)	h (m)	h (m)	h (m)
1.886	0.80	0.46	–	1.28
2.085	0.86	0.47	0.41	1.30

2.759 - 0.52 0.45 1.35

3.876 - 0.60 0.58 1.43

the bed is made of coarse material with typical sizes of about 50 mm (Carlier d'Odeigne et al., submitted). This is illustrated in the grain size distributions, e.g. at the upstream end of the study reach (Dory, blue curve in Figure 4) and at the downstream end (Grand-Place, red curve in Figure 4). This evolution is illustrated in Figure 5. The d 50 diameter along the study reach is shown in Figure 5.

The Cavaillon River has two main tributaries: River Gros-Marin and River Bonne-Fin, as illustrated in Fig ure 1. During the dry periods, no water flows from the tributaries into the Cavaillon River. However, during floods, the contribution of these tributaries will be con sidered by estimating the additional discharge issued from runoff over the corresponding watersheds.

According to previous works (Gonomy, 2012), dis charges up to 400 m 3 /s were observed in the region. However, within the frame of the present project, only very low discharges were measured in the river as the region is experiencing a severe drought in the last 4 years. The data available for the current project are detailed in Table 1: for each measured discharge, the water depths at three stations located along the river were also recorded, yielding a first

database that will be used to calibrate the hydraulic

model. Each station is identified by its kilometer

distance from Dory as PK00.000 (x = 0 km). Sta

tion PK23.016 (x = 23.016 km) corresponds to Grand

Place, the downstream limit of the study reach.

3 HYDRAULIC MODEL

3.1 Bathymetry

A one-dimensional model of the river reach is con

structed based on the collected field data. As can

be observed in Figure 6, abrupt changes in the Figure 6. Distance from the thalweg to the banks at the cross-sections along the Cavaillon River. Figure 7. Dory weir. cross-section width occur in the river, with local adverse slopes (Figure 2). 3.2 Upstream boundary condition The upstream end of the study reach is located at the Dory weir, for which a stage-discharge relations has to be established. Indeed, as can be observed from Figure 7, the weir has a typical Creager shape, but was seriously damaged during previous floods. In particular, the crest level is no more horizontal and the downstream concrete slab was lifted upwards, probably due to important under pressures. As the water level is monitored both at the upstream and downstream side of the weir, one of the objectives of the study is to construct a stage-discharge relationship that could be used to measure the discharge entering the study reach. The stage-discharge relationship can be written as Based on the field measurements reported by Rousseaux (2014), an average discharge coefficient μ= 0.46 could be estimated for the weir. 3.3 Downstream boundary condition The downstream boundary condition of the model is located at Grand Place, i.e. about 4 km from the mouth of the river. At this location, the water level is recorded

Figure 8. Water-level evolution during a tidal cycle

(Grand-Place: x = 23.016 km).

using an OTT PLS device and the discharge is mea sured by an OTT SLD water velocimeter. However, in low-flow conditions, this latter device is no more submerged and no discharge measurements can be obtained.

Because of the proximity of this downstream boundary condition to the mouth of the river, the influ ence of the tidal sea level variations on the water level at this station was investigated. A 24-hour measure ment campaign was carried out to record the water level at Grand Place and at three other stations located further downstream. The results are illustrated in Fig ure 8. While the influence of the tide is clearly visible close to the mouth of the river, its influence at Grand Place appears to be negligible. This should however be confirmed with further measurements for different tidal conditions and rived discharges.

4 MODEL CALIBRATION

4.1 Simulation tool

The steady-flow AxeRiv tool is used, solving the Bernoulli equation considering cross-sections of arbi trary shape. Different levels of interpolation of the flow variables between the cross-sections can be defined, in order to increase the accuracy of the calculated water profile. This feature is particularly useful in the present case, where the distance between the measured cross-sections is about 200m and where the cross section can vary significantly from one section to the other (Figure 2 and Figure 6). Typically, in the present calculations, 10 interpolation points were used.

4.2 Calibration of the bed roughness

Using AxeRiv, flow profiles were calculated for the different measured discharges of Table 1, and the downstream water depth imposed at Grand-Place, i.e. PK23.016 in Table 1. Using the available water depth measurements, an average value of the Manning coefficient for the considered river reach could be cal ibrated as n = 0.030. Such a water profile is illustrated in Figure 9 for a discharge Q = 2.085 m^3/s.

At this stage only an averaged value of the rough ness was estimates.. More refined estimation of the

roughness could be possible, for example using the Figure 9. Water profile – Q = 2.085 m^3/s, n = 0.030. measured median diameter, but the validation of such refined approach requires more field measurements than available at the moment, especially during flood events. 5 FLOOD SCENARIOS The Cavaillon River is located in a region where hydrological extremes are frequent. So, determining flood and inundation risk is one of the main goals of the present study. Most of the classical approaches are not usable in the present case, due to poor availability of topographical data. So, the used methodology aims at providing results accurate enough in a context of inaccurate data. 5.1 Considered discharges No systematic hydrological data are available in Haiti, especially regarding extreme discharges. One of the main issues for the future is the reconstruction of past events in order to determine at least approximate discharges for the evaluation of floods and extent of inundation area. An estimate of 535 m^3/s for the 100-year return period discharge was found in

Gonomy (2012), but the way to go to this value is not known. So, this value was used but, considering that such a value leads to water levels below what was observed in situ, an arbitrary value of 1000 m^3/s was also considered. This high discharge presents the advantage to lead to results more visible on the maps. 5.2 Methodology Some characteristics of Haitian topographical data are challenging for flood scenarios: only a 30 m × 30 m DTM is available, its geo-referencing is not completely clear and the reference elevation is not linked to other data. The following classical steps for evaluating the inundation extent along the Cavaillon River were applied. Firstly, the water profile is computed, using the cross-sections measured every 200 m and assuming a steady-flow. At each section, two points are selected at some distance from the river axis (Figure 10). Assuming the same water level at these points

Figure 10. Example of fixed levels (in yellow) for spline interpolation – Q = 535 m^3/s, n = 0.030 – Section 34 corresponds to x = 7.062 km and section 37 to x = 7.720 km.

Figure 11. Spline interpolated raster surface with points of Figure 10 as basis (each square is a 30 m × 30 m cell). Top: interpolated water table; bottom: interpolated water table with transparency to identify the background photograph.

as the computed one along the river axis; these points form a shapefile that can be used in GIS software (in the present study the free software QGIS was used). These points form a basis for a spline interpolated raster surface (Figure 11) that could represent approximately the flood water table, even in river bends (in the present study the SAGA tool box was used in combination with QGIS).

The spline interpolated raster surface may be cre ated with any density but, as the result has to be com

pared to the 30 m × 30 m DTM raster, the same density

has been adopted as can be observed in Figure 11.

The spline interpolated surface is now compared

with the DTM and, in case of positive difference of

water table and terrain levels, a new raster is created

representing the water depth at the different raster cells

(Figure 12). Once again the resolution of such a map

is the same as the DTM one, leading to results that

are to be interpreted because, among others, of the

coarseness of the grid. Figure 12. Water depth from the positive difference between spline interpolated raster surface and DTM (each square represents a 30 m × 30 m cell). Figure 13. Water depth – Q = 535 m^3/s. Figure 14. Water depth – Q = 1000 m^3/s. 5.3 Results and discussion The procedure described before was applied along a part of the Cavaillon River for discharges Q of 535 and 1000 m^3/s, respectively. The resulting water-depth maps are presented in Figures 13 and 14. Considering as well large scale maps of Figures 13 and 14 as detailed map of Figure 12, some points of discussion clearly arise. As expected the flood extent is considerably larger for the largest discharge and the water depth is significantly increased. However, even if flood areas are generally located along the river axis (materialized by a red line in Figures 13 and 14), some appear outside the river path, while the river itself appears as emerged. This is clearly the case for sections 35 to 37 in Figure 12 that are unsubmerged while a rather deep

Figure 15. Profile across section 35: the red curve repre

sents the DTM profile, the blue curve is the section across

the spline interpolated surface (the water depth color bar is

the same as in Figure 12 but some transparency was allowed to the water table layer, so altering the colors). inundation occurs at some distance of the southern bank of the river.

It could be suspected that something was wrong in the process, but an extended profile across section 35 (Figure 15) shows that the flood area is perfectly in accordance with the DTM and water table levels. According to this profile, the river bed would be higher than the water level, which is impossible in case of flood, and the lowest point of the terrain would be out side of the river path. It is interesting to observe that the water table (the blue profile in Figure 15) is practi cally horizontal, so indicating that the always possible distortion of the spline interpolation does not affect too much the water table even in sinuous river path.

Since the 30 m × 30 m DTM is not really docu mented, some discrepancy might exist between this DTM and the real situation: the river seems rather instable in the considered reach and the depression along the southern bank of the river could be an old course of the river, which is partly suggested by the Google satellite view in Figures 12 and 15, where a sinuous path may be observed near the depression.

On the other hand, the fact that the river seems unsubmerged at some place during flood events could be due to the too coarse available DTM. Consider

ing Figures 12 and 15, where each square of the water

depth map represents a 30 m × 30 m cell, it is clear that

the minor bed of the river is often less than 30-m wide,

in such a way that the DTM may miss the actual thal

weg of the river. Moreover it is probable that the DTM

was built without accounting for the submerged part

of the river, so artificially erasing the river bathymetry.

Another explanation could be found in the fact that

the reference level of the DTM is uncertain, so open

ing the possibility that the reference level used for the

Simulating runoff generation and consumption for rehabilitation

of downstream ecosystems in arid northwest China

C. He

Key Laboratory of West China's Environmental System (Ministry of Education), College of Earth and

Environmental Sciences, Lanzhou University, Lanzhou, China

Department of Geography, Western Michigan University, Kalamazoo, Michigan, USA

L. Zhang

Key Laboratory of West China's Environmental System (Ministry of Education), College of Earth and

Environmental Sciences, Lanzhou University, Lanzhou, China

ABSTRACT

Proper allocation of the limited water resources among

competing uses is essential to ensure the welfare of

human beings and the sustainability of ecosystems, especially in arid regions such as Northwest China. The Heihe River Watershed is the second largest inland river (terminal lake) in China, with a drainage area of 128,000 km 2 . From the headwaters in the south to the lower reach in the north, the Heihe River Water shed physically consists of the Qilian Mountain, the Hexi Corridor, and the Alashan Highland. The Qilian Mountain is situated at the south of the watershed, with a peak elevation of 5,584 m. Located in the middle reach of the Heihe River Watershed, the Hexi Cor ridor hosts over 90% of the total agricultural oases in the watershed and supports more than 97 percent of the Heihe watershed's nearly 2 million inhibits. North of the Hexi Corridor is the Alashan Highland, an extremely dry desert with an annual precipitation below 50 mm. Since the 1970s, the increased with drawals for agricultural irrigation in the Hexi Corridor have depleted much of the river flows to the lower reach, endangering aquatic ecosystems, accelerating desertification, intensifying water conflicts between the middle reach and lower reach users. To mitigate the water conflicts, the State Council of China has issued a "Water Allocation Plan for the Heihe Water shed Mainstream", mandating the allocation of 0.95

billion (10^9) m^3 of water annually to the lower reach

under normal climatic conditions for rehabilitation of

downstream ecosystems. However, are the flows from

the upper and middle reaches sufficient to deliver 0.95

billion (10^9) m^3 of the water downstream annually?

This paper adapted the Distributed Large Basin

Runoff Model (DLBRM) to the Heihe River Water

shed to gain an understanding of the generation of glacial/snow melt, surface runoff and groundwater in the mountainous upper reach, and distribution of evapotranspiration (consumptive water use) in the middle reach of the watershed. The DLBRM was calibrated over the period of 1978–1987 (a wet hydrologic period) for each of the 9,790 cells (cell size: 4-km^2) at daily intervals. The calibration shows a 0.696 correlation between simulated and observed watershed outflows. The ratio of model to actual mean flow was 1.023. Over a separate simulation period (1990–2000, a normal hydrologic period), the model demonstrated a 0.717 correlation between simulated and observed watershed outflows, and the ratio of model to actual mean flow was 1.069. Simulation of the daily river flows for the period of 1990–2000 by the DLBRM shows that Qilian Mountain in the upper reach produced most of the runoff in the watershed. Annually, the simulated average annual flow for 1990–2000 was about 0.896×10^9 m^3 from the middle reach to the lower reach under a normal, median precipitation year (P = 50%), which falls short to meet the requirement of delivering 0.95×10^9 m^3 downstream annually mandated by the State Council 50 percent of the time. Under drier climatic conditions, even less amount of flow would be delivered downstream, posing an even greater challenge for restoring downstream ecosystem services. To tackle the increasing water conflicts among the upper, middle, and downstream users, we suggest that stakeholders from different levels of governmental agencies and private institutions be fully engaged in the watershed management process to develop a water allocation system that consists of multiple water allocation criteria, implementation plan, evaluation and feedback mechanisms. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Integrating Spatial Multi Criteria Decision Making (SMCDM) with

Geographic Information Systems (GIS) for determining the most suitable

areas for artificial groundwater recharge

M. Zare & M. Koch

Department of Geotechnology and Geohydraulics, University of Kassel, Kassel, Germany

ABSTRACT: The Shabestar plain is located in northwest of Iran. Increasing population, agricultural devel

opment and illegal well pumping have increased the exploitation of groundwater resources in this plain. This

phenomenon, along with recent droughts, has led to severe decreases of the groundwater levels over the last

years. In order to reduce this crisis, the establishment of groundwater artificial recharge projects can be a suit

able solution. An important step in the realization of such projects is determining suitable areas. In this study

the Spatial Multi Criteria Decision Making (SMCDM) was used in conjunction with Geographic Information

Systems (GIS) for that purpose. More specifically, seven main parameters including land slope, soil hydrologic

groups, alluvium thickness, quaternary units, groundwater level were considered as the main layers in GIS, while

pasture land and water drainage network were used as efficient layers for locating artificial recharge areas. The

data layers for each one of these variables were supplied by GIS. The ranges of change of the five main layers

were then classified in accordance with their importance in locating process. These data layers were afterwards

assessed with one another by means of a pairwise comparison matrix with regard to their significance to locating

by applying the Analytical Hierarchy System (AHP)

technique. The selected areas are integrated with exclusion

ary areas from pasture lands and the presence of water drainage networks. Finally seven separate regions with

an area of 124.3 km 2 (=10.42% of the flood plain) are identified as appropriate flood spreading recharge areas.

Based on the annual potential of runoff production, that has been calculated by Justin's method, the areas were

prioritized. Thus region #3, with a surface of 38.6 km 2 turns out to be best place for artificial recharge, as it has

7.65 million cubic meter (MCM) runoff production per year, whereas regions #2 and #1 have second and third

priority, respectively.

1 INTRODUCTION

Changing hydrological conditions occurring, for example, in the wake of future climate change (IPCC, 2007) by alterations of temperatures and precipita tion, will have detrimental effects on the surface and groundwater resources in many areas of the world (e.g. Koch, 2008; Fink and Koch, 2010). This holds par ticularly for regions and countries which are already nowadays affected by water scarcity, such as the Mid dle Eastern region, including Iran. There, responding also to the needs of a strongly increasing population, rising water withdrawals have already caused drastic changes in the surface flow regimes and severe drops in groundwater levels in many watersheds of that country (Zare and Koch, 2014).

Because of Iran's location in an arid and semi-arid climate region, groundwater is a major water resource for many areas of that country. One efficient way to overcome deficits in groundwater resources consists in artificial recharge of groundwater (Zare and Taheri, 2011), for example, by a flood spreading approach.

The main goals of artificial groundwater recharge are improving the water quality, adding or maintaining groundwater as an economic resource, preventing saltwater intrusion, and reducing or even preventing land subsidence. Thus artificial recharge projects provide mechanisms to protect groundwater in general terms. The problem is then to locate suitable areas for appropriate artificial recharge projects. Many studies to that avail have been done which are mainly based on Remote Sensingand Geographic Information System (GIS) – techniques (Ravi Shankar and Mohan 2005, Saravi et al. 2006, Ghayoumian et al. 2007 and Alesheikh et al. 2008, Arlai et al. 2010, Koch and Sirhan 2012, Sirhan and Koch 2014,). Apart from using mathematical models for artificial recharge modeling, using spatial data in GIS for the identification of potential areas for artificial recharge have become very common (Sharma 1992, Sukumar and Sankar 2010). GIS is a computerized data management system to capture, store, manage, retrieve, analyse, and display spatial information. GIS reduces time and costs for the selection of sites and provides a digital data bank for a future monitoring program of these selected sites (Al-Adamat, 2012).

Classical GIS has been used by Saraf and Chadhury (1998) to produce suitable areas map for ground water recharging in Madhya Pradesh area of India using information from satellite images and other layers, such as land use, cover vegetation, geomor phology, and geology. Patil and Mohite (2014) deter mined potential groundwater recharge zones in the Pune district of Maharashtra, India. They consid ered the geomorphology, soil, land use land cover, slope (%), drainage density and lineament density, which were prepared using satellite imagery and other conventional data. The thematic layers were first digitized from satellite imagery, then all the matic layers were integrated using ArcGIS software to identify the groundwater recharge potential zones for the study area to generate a map showing these groundwater recharge potential zones, namely, ‘poorly suitable’, ‘moderately suitable’ and ‘most suitable’ based on knowledge-based weighting factors. There have been various other improvements in the normal GIS approach to incorporate some kind of quantita tive decision making factors for suitable site selection. Thus Zehtabian et al. (2001) incorporated a fuzzy logic approach into the GIS and so determined suitable areas for flood spreading in the Toghrodd Watershed in the Ghom province, Iran. Raid et al. (2011) presented two approaches for this kind of analysis: the classical one where a suitability map to find out the suitability of every location on the map was created and a more sophisticated one which consisted in querying the cre ated data sets to obtain a Boolean result of true or false map.These techniques have been applied to Sadat Industrial City in the Nile delta, Egypt. Thematic lay ers for a number of parameters were prepared from some maps and satellite images and they were classi fied, weighted and integrated in ArcGIS environment. By means of the overlay weighted model in ArcGIS, a suitability map which is classified into number of pri ority zones was obtained and could be compared with the obtained true-false map from the Boolean logic. Both methods suggested mostly the northern parts of the city to be suitable for groundwater recharge, how ever, the weighted model provided a more accurate suitability map while the Boolean logic suggested a wider range of areas.

A further refinement of the classical GIS map ping approach for suitable site selection consists in the incorporation of MCDM (Multi Criteria Decision Making), the latter being a sub-discipline of operations research that explicitly considers multiple criteria in decision-making environments. Although taking GIS and MCDM alone will not effectively facilitate the implementation of site selection project parameters which are equally based on complex decision criteria and spatial information (Jun, 2000) and, so may lead to a poor decision (Bailey et al. 2003), MCDM integrated into GIS (SMCDM) provides more adequate solution procedures to this problem as the selection of suitable areas for artificial recharge projects may be done more

comprehensively (Calijuri et al. 2004). MCDM can

be used to evaluate standard multi-criteria problems with a set of alternatives. In standard MCDM subjects are not spatial, that is, the impact of an alternative for one criterion is a value. Although there is a variety of techniques for the determination of the criteria weights, one of the most famous methods developed by Saaty (1977) consists in doing pairwise comparisons. This technique is known as the Analytical Hierarchy Process (AHP) which is one of the most versatile techniques of MCDM. AHP uses multiple variables for the decision making, and can be employed to formulate the problem and to solve it hierarchically (Nagarlu et al. 2012). In this study, Spatial MCDM (SMCDM) in conjunction with Geographic Information Systems (GIS) will be used to locate suitable areas for artificial recharge in the Shabestar plain in northwest of Iran. For this purpose, seven main parameters, including land slope, soil hydrologic groups, alluvium thickness, quaternary units, groundwater level are considered as main layers, while pasture land and water drainage network are considered as efficient layers. 2 MATERIALS AND METHODES 2.1 Study area The Shabestar plain is located in northwest of Iran (Fig. 1). The plain's area is 1200 km 2 , while its elevation varies between 1266 m above sea level in the south and 3100 m in the north. The average annual rainfall is about 248 mm. Based on the available data, seven effective parameters, including land slope, soil hydrologic groups, alluvium thickness, quaternary units, groundwater level are considered as main layers, while pasture land and water drainage network are used as efficient layers for locating appropriate artificial recharge areas that could be supplied by means of flood spreading. Figure 1. The Shabestar plain in northwest Iran.

2.2 Selection of main criteria (data layers) for artificial recharge

Based on the available data, seven effective parame

ters, including land slope, soil hydrologic groups, allu

vium thickness, quaternary units, groundwater level

are considered as main layers, while pasture land and water drainage network are used as efficient layers for locating appropriate artificial recharge areas that could be supplied by means of flood spreading. The DEM (Digital Elevation Model), groundwater data, soil and geology maps, and pasture land data were provide by the Water Organization of the Azarbaijan-e-Sharghi province.

2.3 Data layer preparation in GIS

Slope is a fundamental contributing factor in the selec tion of flood spreading areas, as water velocity is directly related to the land slope. On steep slopes, runoff is more erosive and can easily remove detached sediments and transport it down the slope (Faucette et al. 2003). The experience gained from the analysis of 36 flood spreading areas across Iran indicates that the most suitable areas for flood spreading should have a slope less than 5 % (Alesheikh et al. 2008).

For generating the slope layer, the Arc Hydro model has been used in the Arc GIS environment. In this regard, the DEM is modified by the Arc Hydro model and then the slope layer is generated (Fig. 2). From the figure one can observe that most of the southern area of the plain is almost flat, whereas in the northern part, due to the hills, the slope is about 97%. To sum up, the slopes are ranging from 0% in the south to 97% in the north.

Quaternary sediments, which are the most impor tant reservoir of groundwater resources, have been formed from physical and mechanical weathering of upland formations. Thus the geological properties of these quaternary sediments are instrumental for the characterization of the hydrology and hydrogeology of the water in the groundwater reservoir. In general such sedimentary structures have flood storage poten tial. Quaternary sediments are very important in Iran because they cover more than 50% of the surface of the country (Chenini et al. 2010, Vorhauser and Ham lett 1996). Fig. 3 shows that more than 50% of the Shabestar plain’s area is covered by quaternary units that, basically, are suitable for flood spreading. How ever, for effective recharge the soil material in the porous media must have, in addition, a high vertical permeability to aid infiltration and the aquifer must be sufficiently transmissive to transport the water away from the spreading area. The importance of infiltra tion when using the flood spreading method comes from the fact that a low infiltration rate would restrict the entrance of water into the porous media (Liu et al. 2001). As this restriction often occurs at the soil sur face, the quantification of the infiltration rate is crucial for the evaluation of the efficiency of this kind of

groundwater recharge method. Figure 2. Left.panel: DEM of the Shabestar plain; right panel: Slopes of the study area Figure 3. Left panel: Quaternary units of the Shabestar plain; right panel: HSG layer of the study area. The most important parameters influencing the infiltration rate are the physical characteristics of the soil and the vegetation cover on the soil surface. For the quantification of the infiltration parameters the soils are classified into four hydrologic soil groups (HSG's) – namely A, B, C and D – to indicate the minimum rate of infiltration obtained for a bare soil after prolonged wetting. The infiltration rate is then controlled by the surface conditions. HSG also indicates the transmission rate, the rate at which the water moves within the soil. This rate is controlled by the soil profile. Approximate numerical ranges for the transmission rates as defined in the HSG groups were first published by Musgrave (USDA 1955). The transmission rate in group A – which is the best for artificial recharge because of high infiltration rate – is greater than 0.76 cm/hr, whereas the rates in group B, C and D are 0.38–0.76 cm/hr, 0.13–0.38 cm/hr and <0.13 cm/hr, respectively. The spatial distribution of HSG is shown in Figure 3. For flood spreading the groundwater table should be sufficiently deep so that adequate storage space in the vadose zone is available to accommodate the recharge water. In places where the depth of the groundwater table is less than 3 to 4 m, artificial recharge should not be considered, as there is the danger of water logging that causes tremendous economic and environmental losses, because of, among other factors, increasing soil salinity (Zare and Koch, 2014). Alluvial thickness is another factor that affects the selection of an area for artificial recharge. A suitable site for flood spreading is a place with thick alluvial fans. If all factors are suitable, except the alluvial thickness, artificial recharge may cause saturation of the recharged layer and water logging may occur (Ghayoumian et al. 2005, Zare and Koch, 2014). In order to generate the alluvium thickness layer, 42 observation log wells have been used. 10 extra wells are then used to validate the

Figure 4. Left.panel: Alluvium thickness of the Shabestar

plain; right panel: Groundwater level layer of the study area.

results of the spatial interpolation. For the groundwater

level map piezometric well data observed in January

2009 have been used. For both maps spatial interpo lation is employed which creates a continuous surface (Raster map) from the measured point data (Schaben berger and Gotway, 2005). Three methods of spatial interpolations are practiced in this research, namely, (a) Ordinary Kriging with exponential-, sphericaland Gaussian variogram models, (b) Spline (Theissen and regularized method) and (c) inverse distance weight ing (IDW). The mean absolute error (MAE) is used to identify the goodness of fit of the interpolation. Thus the most appropriate method should have the least MAE, wherefore the MAE can range from 0 to ∞.

The interpolation exercise shows the Kriging method with a spherical variogram to be the most accurate method for both parameters, with MAE = 0.18 and 0.21 for groundwater levels and alluvial thicknesses, respectively.

As Fig. 4 shows, the alluvium thicknesses range from 33 to 145 meters and groundwater levels between 4 and 57 meters. Concerning their importance in arti ficial recharge, these two parameters will later during theAHP-process be classified into four categories with specific weights (see Table 4).

2.4 SMCDM using AHP

MCDM is a sub-discipline of operations research that explicitly considers multiple criteria in decision making environments. MCDM can be used to evaluate standard multi-criteria problems with a set of alter natives. In standard multi-criteria decision methods subjects are not spatial, that is, the impact of an alterna tive for one criterion is a value.The SMSDM stands for an integration of spatial analyses and MCDM (Fig. 5). One of the MCDM methods is the Analytic Hier archy Process (AHP) that was introduced by Saaty (1977) and has become a very popular means since then to calculate the needed weighting factors by help of a preference matrix where all identified rel evant criteria are compared against each other with reproducible preference factors. In short, MCDM is a method to derive ratio scales from paired comparisons. The input can be obtained from actual measurements such as price, weight etc., or from subjective opin ion such as satisfaction feelings and preference. AHP allows for some small inconsistency in the judgment, because human opinion is not always consistent. The Figure 5. Integration of spatial analysis and multicriteria methods into spatial multicriteria methods. ratio scales are derived from the principal eigenvectors and the consistency index from the principal eigenvalue of the pair-wise comparison matrix. All criteria/factors which are considered relevant for a decision are compared against each other in a pair-wise comparison matrix which is a measure to express the relative preference among these factors. Therefore, numerical values expressing a judgement of the relative importance (or preference) of one factor against another have to be assigned to each factor. Since it is known from psychological studies that an individual cannot simultaneously compare more than 7 ± 2 elements, Saaty (1977) and Saaty and Vargas (1991) suggested a scale for comparison, consisting of integer values between 1 and 9 which describe the intensity of importance (preference/dominance). This a value of 1 expresses “equal importance” and a value of 9 is given for those factors having an “extreme importance” compared with the other ones (Table 1). In the present study, the eigenvector method of the AHP algorithm has been used in the Arc GIS environment for the weighting and the pair-wise scoring of the spatial layers. For the evaluation of the main layers, the local conditions of the region, the relevant literature and the specialist expertise (15 questionnaire forms were prepared and filled out by hydrogeologists and geologists that have sufficient information about the study area) were considered. Based on this information, the 5 by 5 pair-wise comparison matrix (reciprocal matrix) A for the 5 criteria as shown in Table 2 has been generated. For example, the value of “5” in the HSG row/Alluvial thickness column means that HSG is “strongly” important compared with alluvial thickness. The next step in the AHPmethod is then the computations of the effective weights w_i for each of the main indicator layers of Table 2. Practically, the most common approach to do this consists in taking as weights the normalized components of the eigenvector associated with the largest eigenvalue of the pair-wise comparison matrix A (Gao et al, 2009). Thus the following eigenvalue problem is solved: Where λ denotes the eigenvalues of A and W are the associated eigenvectors. The weights w_i, equal to the

Table 1. Comparison scales (Saaty & Vargas 1991).

Intensity of importance	Description
1	Equal importance
3	Moderate importance of one factor over another
5	Strong or essential importance
7	Very strong importance
9	Extreme importance
2,4,6,8	Intermediate values

Table 2. Pair-wise comparison matrix (reciprocal).

Criteria	Slope	HSG	Quaternary units	Alluvial thickness	Groundwater level
Slope	1	3	5	7	9
HSG	1/3	1	2	5	9
Quaternary units	1/5	1/2	1	3	7
Alluvial thickness	1/7	1/5	1/3	1	3
Groundwater level	1/9	1/9	1/7	1/3	1
Sum (S_i)	1.79	4.81	8.48	16.33	29

Table 3. Calculated weights by the AHP method.

Criteria	Slope	HSG	Quaternary units	Alluvial thickness	Groundwater level
Weights (w_i)	0.51	0.25	0.15	0.06	0.03

normalized components of the eigenvector associated with the largest eigenvalue λ_{max} , are listed in Table 3. Of course, the values of the pair-wise comparison matrix will normally be well specified and are not set remain inconsistent and intransitive and may then lead to variations in the eigenvector calculations, i.e. different weights. Such inconsistencies might be of the form that a criteria Ai, being preferred over another criteria Aj , with Aj being preferred over a criteria Ak is not preferred overAk (Ai must be preferred overAk in this case). Therefore, Saaty (1977) proposed a consistency

ratio (CR) which is a single numerical index to check

the pair-wise comparison matrix for consistency. CR

is defined as the ratio of the consistency index CI to

the average consistency index RI:

and RI is the random consistency index that depends

on n, the number of criteria. A list of RI as a func

tion of n has been provided by Saaty (1977) and is Table 4. Random consistency index RI for different n (Saaty, 1977). n 3 4 5 6 7 8 9 10 RI 0.58 0.9 1.12 1.24 1.32 1.41 1.45 1.49 Table 5. The main layers and sub-layers weights in AHP modeling. Weight Weights Criteria (Layers) (Layers) Sub-layers (sub-layers) Slope (%) 0.51 0–5 0.949 5–8 0.050 >8 0.001 HSG 0.25 A 0.527 B 0.315 C 0.149 D 0.009 Alluvial thickness 0.06 0–10 0.010 (m) 10–40 0.143 40–80 0.293 >80 0.562 Groundwater level 0.03 0–10 0.009 (m) 10–20 0.567 20–30 0.315 >30 0.110 Quaternary units 0.15 Q t1 0.580 Q t2 0.289 Q sf 0.115 Non Q 0.016 listed in Table 4. Thus, with n=5 as in the present application, RI = 1.12. With Eq. (2) this results in CR = 0.052, which is less than 0.1, which is the critical CR value to accept the pair-wise comparison matrix, defined in Table 2. Regarding the importance of each criteria in the flood spreading method, all 5 parameter (Table 3) maps have been further classified into specific sub-layers. Each sub-layer has been scored following the results of the questionnaire form, the statistical analyses and the literature review for the same artificial recharge projects in semi-arid regions of Iran (Zehtabian et al. 2001, Taheri and Zare 2013, Ghayoumian et al. 2007). Table 5 shows the layers and sub-layers weights obtained by this procedure in the AHP modeling. Based on these 5 main spatial layers, the suitable areas for artificial recharge have been determined by the AHP extension in the Arc GIS software environment. As Fig.6 shows, all the suitable areas are located in the southern part of the plain. 2.5 Integration with pasture land and drainage network data layers The areas which are not suitable for groundwater recharge by flood spreading are called exclusionary

Figure 6. Left panel: Artificial recharge possibility map

obtained with the AHP method; Right panel: Pasture land.

Figure 7. Left panel: Suitable recharge areas; right panel: Combination of suitable areas with drainage network.

areas. In the present study, cultivation farms, gar dens, private lands – due to social tensionsand urban areas were considered as exclusionary areas for the flood spreading operation. Once the suitable areas have been determined by theAHP process, the exclusionary areas from pasture land are deducted from the suitable recharge areas map in Arc GIS environment and seven areas have been determined.

Flood spreading in possible recharge areas is not possible without a drainage network, due to the lack of runoff production in upstream region which acts as a source of recharge. For the generation of the drainage network layer, the DEM in the Arc Hydro model is applied. Fig.6 and 7 show that the selected 7 areas are not restricted in this respect.

2.6 Potential of run-off production

Although estimating annual runoff of rivers that have hydrometric stations is simple, it is complicated in watersheds that have not enough measured data. For quantifying the potential of runoff production in such watersheds, the Justin method (Justin, 1914) has often been used (Alizadeh, 2005). The latter is based on the idea that similar watersheds have also similar runoff

characteristics. The first step in the Justin method

starts then with the computation of the following three

parameters for the reference watershed with enough

measureable data:

Where H_{max}, H_{min} are the maximum and minimum

elevations of the watershed (km), A is its area (km^2),

W is the annual runoff volume (MCM),T is the average Table 6. Justin's K-calculation for the Daryan-chay watershed. A H max H min W P T (km 2) (km) (km) S (MCM) (cm) (∘ C) K 53 2.749 1.226 0.203 15.39 24.84 12.5 0.032822 Table 7. Computation of potential annual runoff production using Justin's K for the seven selected recharge areas (see Fig. 7). Selected A H max H min P T W area # (km 2) (km) (km) S (cm) (∘ C) (MCM) 1 21.64 1.32 1.28 0.009 24.84 12.5 3.86 2 33.67 1.34 1.31 0.006 24.84 12.5 5.64 3 38.63 1.44 1.33 0.017 24.84 12.5 7.65 4 5.24 1.50 1.34 0.068 24.84 12.5 1.28 5 7.25 1.65 1.49 0.059 24.84 12.5 1.74 6 9.00 1.40 1.32 0.027 24.84 12.5 1.91 7 8.89 1.54 1.34 0.063 24.84 12.5 2.15 temperature (∘ C), P is the annual precipitation (cm) and S is the average slope. With all this known data in Daryan-chay watershed Eq.(6) defines the Justin coefficient K. In this study, the Daryan-chay watershed is taken as the reference watershed, as it has enough measured data. The mean annual discharge of Daryan-chay is 0.49 m3/s that is equal to W = 15.39 MCM. Using the other topological and meteorological data for that the watershed as indicated inTable 6, Justin's K coefficient is calculated as 0.0328. In the second step, using the K- coefficient for the reference watershed, Eqs. (4) to (6) are solved in reverse order for the annual runoffs W for each of the 7 selected areas indicated in Fig. 7, using the meteorological variables from the reference watershed. The results are listed in Table 7. 3 RESULTS AND DISSCUSSIONS 3.1 Discussion of the physical parameters of the watershed defining suitable artificial recharge areas Regarding the slope parameter shown already coarsely in Fig. 2, the more detailed map of Fig. 8 indicates the slopes of the south and central areas of the Shabestar plain are less than 5% that is suitable for groundwater recharge by flood spreading. The groundwater level spatial map in Fig.8 shows that the groundwater depths in most parts of the study area range between 10-20 or 20-30 m, that is suitable for groundwater recharge. The alluvium thickness ranges in areas that have suitable slopes are between 30 and 80 meters which, in turn, are sufficient for artificial recharge projects.

Figure 8. Left panel: Reclassified slope layer; right panel:

Groundwater level categories in the study area.

Table 8. Results of the five-layer integration by the AHP

method.

Artificial recharge potential	A (km^2)	A (%)
suitable	505	44.93
fairly suitable	49	4.36
not suitable	103	9.16
impossible	467	41.55

Fig. 3 shows that the south and central parts of the plain are composed of young alluviums called quaternary, so that flood spreading is possible in these areas.For the evaluation of the important characteristics of infil tration, the Hydrologic Soil Group (HSG) has been studied. As Fig.2 shows, in flat areas (slope <5%) of the study region the HSG is A or B, i.e. these are good for flood spreading projects.

Based on the five data layers that have been integrated by the eigenvector method in the AHP procedure (Table 4), a final classification of the suit ability areas for artificial recharge has been made. The results are listed in Table 8 and show that about 550 km^2 , corresponding to 50% of the total study area, are prone for recharge.

The spatial distribution of the suitable recharge

areas listed in Table 3 reveals that the slope is the most

important criterion in the evaluation of the suitabil

ity for flood spreading recharge. However, one must

al-so take into consideration so-called exclusion areas

such as pasture lands as well the water drainage net

work, both of which play an important role in ar-tificial

recharge site selection. In fact, the flood spreading

method could not be performed on private farm lands

and the drainage line is necessary for driving the runoff

to the project area. This last crite-rion is eliminating

project areas smaller than 10 ha because the flood

spreading method in small areas is not possible. Sub

tracting these small areas from the suitable ones of

Table 8 results, finally, in seven sep-arate regions with

a total area of 124.32 km 2 (~10.42% of the flood plain

area) that are well suitable for the application of a flood

spreading artificial re-charge method.

3.2 Prioritizing the selected areas according to the potential of runoff production

Once the suitable areas have been chosen, it is impor

tant to know which area is the best for establishing Table 9. Prioritizing selected recharge areas based on annual runoff. Selected area no. A (km 2) W(MCM) Priority 1 21.64 3.86 3 rd 2 33.67 5.64 2 nd 3 38.63 7.65 1 st 4 5.24 1.28 7 th 5 7.25 1.74 6 th 6 9 1.91 5 th 7 8.89 2.15 4 th a groundwater recharge project, because for flood spreading some additional factors should be considered, such as possible soil erosion which may reduce the infiltration due to sediment settling, and vegetation plant implementation which, in reverse, may moderate this adverse effect again. In this regard, preparing an area for flood spreading has some costs, so it is better to perform it in an area which has more water available. For prioritizing the selected suitable recharge areas, the potential of annual runoff production is considered as a final factor. The former is computed by means of the Justin method (see Table 7). Based on these results the region number 3 (see Fig. 8) turns out to be the most suitable area for artificial groundwater recharge (Table 9). After determination of the priority of the recharge areas, the most appropriate time period for the application of the artificial recharge should be determined. According to the long-term statistics, the maximum discharge of flood occurs during months of February, March andApril, so that during these months the probability of floods is high while, at the same time, the agricultural use of water is low. As, in addition, the humidity is high in this season, the rate of evaporation is also low. This means that artificial groundwater recharge is most appropriately effected during these early months of a year. 4 CONCLUSIONS In the process of studying artificial recharge projects the determination of the most suitable areas for flood spreading is very important. Geographic Information System (GIS) as an efficient, fast, accurate and inexpensive, highly-efficient tool has been used in this study. Decision making using the AHP-method for evaluating the accuracy of the weights and of different topographic, hydrographic and geological criteria is very valuable, because this method provides a logic and scientific-based approach to make quantitative comparisons of the different criteria-variables. In this study, the Spatial Multi Criteria Decision Making (SMCDM) has been used within an Arc GIS environment for determining the most suitable areas of artificial recharge in the Shabestar plain in northwest Iran. Five important criteria-parameters determining

most likely artificial recharge, namely slope, HSG,

quaternary units, alluvium thickness and groundwater

levels were overlaid by the AHP-method and, apply

ing an eigenvector analysis of the comparison matrix,

the corresponding weights for the various criteria are

computed. The results indicate that the slope is the most important parameter determining the suitability of an area for artificial recharge. The suitable areas found in this way are further processed, to take into account pasture lands and the presence of a minimum of a drainage network (exclusionary areas). Even tually, seven separate areas with a total surface of 124.3 km^2 (=10.42% of the flood plain) are selected as the most appropriate places for artificial recharge by a flood spreading method. These selected areas are then prioritized, based on the general availability of water, namely the annual potential of runoff produc tion. Because of a lack of hydrographic information for these areas, Justin's method has been used to that avail, employing hydrographic information from a similar, neighbouring watershed with similar runoff character istics. The final results indicate that region #3 with an area of 38.63 km^2 and an annual runoff of 7.65 MCM is best suited to artificial recharge, followed by regions #2 and #1.

Al-Adamat, R., 2012. The Use of GIS and Google Earth for Preliminary Site Selection of Groundwater Recharge in the Azraq Oasis Area-Jordan. Journal of Water Resource and Protection, 4, 395–399.

Alesheikh, A. A., Soltani, M. J., Nouri, N., and Khalilzadeh, M., 2008. Land assessment for flood spreading site selection using geospatial information system. Interna tional Journal of Environmental Science Technology, 5, 4, 455–462.

Alizadeh, A., 2005. Principles of Applied Hydrology. 18 th edition, University of Imam Reza, Mashhad, Iran. 815 p.

Arlai, P., M. Koch, M. Yongprawat, S. Pirom and T. Kit paisalsakul, 2010. Numerical Investigation of combined Flood Mitigation and Groundwater Recharge in the Chao Phraya River Basin , 17th Congress of theAsia and Pacific Division of the International Association of Hydraulic Engineering and Research (IAHR-APD 2010, Auckland, New Zealand, February 21–24, 2010.

Bailey, D., Goonetilleke,A., Campbell, D., 2003.A new fuzzy multi-criteria evaluation method for group site selection in GIS. Journal of Multi-Criteria Decision Analysis, 12 (6), 1–11.

Calijuri, M.L., Marques, E.T., Lorentez, J.F., Azevedo, R.F.,

Carvalho, C.B., 2004. Multicriteria analysis for the iden tification of waste disposal areas. Geotechnical and Geological Engineering, 22, 299–312.

Chenini, I., Mammou, A. B. and May, M. E., 2010. Groundwater Recharge Zone Mapping Using GIS-Based Multi-Criteria Analysis: A Case Study in Central Tunisia

(Maknassy Basin). Water Resources Management, 24, 5,

921-939.

Gao, S., Zhang, Z. and Cao, C. 2009. New Methods of

Estimating Weights in AHP. International Symposium on

Information Processing. Huangshan, P. R. China, August

21-23, 2009, pp. 201-204. Ghayoumian, J, Mohseni Saravi, M., Feiznia, S. and Nourozi, A. A., 2005. Integrating GIS and DSS for identification of suitable areas for artificial recharge, case study Meimeh Basin, Isfahan, Iran. Journal of Environmental Geology, 47, 493-500 Ghayoumian, J, Ghermezcheshmeh, B., Feiznia, S., Nouri, B., Malekian, A., 2007. Application of GIS techniques to determine areas most suitable for artificial groundwater recharge in a coastal aquifer in southern Iran. Journal of Asian Earth Sciences, 30, 364-374 Faucette, B.; King, W.; Germishuizen, P., 2003. Compost based erosion and sediment control demonstrations. BioCycle, 44, 10, 32-40. Fink, G. and Koch, M, 2010. Climate Change Effects on the Water Balance in the Fulda Catchment, Germany, during the 21st Century, In: Proceedings of the Symposium on "Sustainable Water Resources Management and Climate Change Adaptation", Nakhon Pathom University, Thailand, June 16-17, 2010 IPCC (2007) Climate Change 2007: The Physical Science Basis. Contribution of Working Group I to the Fourth Assessment Report of the Intergovernmental Panel on Climate Change [Solomon, S. et al. (eds.)]. Cambridge University Press, Cambridge, UK. Justin, J. D., 1914. Derivation of Run-Off from Rainfall Data, Vol. LXXVII, ASCE, Reston, VA, 346-363. Jun, C., 2000. Design of an intelligent geographic information system for multicriteria site analysis. URISA Journal, 12, 3, 5-17. Koch, M., 2008. Challenges for future sustainable water resources management in the face of climate change, The 1st NPRU Academic Conference, Nakhon Pathom University, Thailand, October 23-24, 2008. Koch, M. and Sirhan, H., 2012. Development of a 3DGroundwater Model for the Gaza Coastal Aquifer, South Palestine, to investigate the Feasibility of artificial Recharge by Waste Water , Nakhon Pathom University, Nakhon Pathom, Thailand, December 26, 2012. Liu, C.W.; Chen, S. K., Joe, S.W.; Kuo, S. F., 2001. Estimation of the infiltration rate of a paddy field inYen-Lin, Taiwan. Agriculture System, 68, 1, 41-54. Negarlu, D., Nassery, H. R. andAdinehvandi. R, 2012. Determine Suitable Sites for Artificial Recharge Using Hierarchical Analysis (AHP), Remote Sensing (RS) and Geographic Information System (GIS). International Journal of Earth Sciences and Engineering, 5, 1, 1328-1335. Patil, S.G. and Mohit, N. M, 2014. Identification of groundwater recharge potential zones for a watershed using remote sensing and GIS. International Journal of Geomatics and Geosciences, 4, 3, 485–493. Ravi Shankar, M. N., and Mohan, G., 2005. A GIS based hydrogeomorphic approach for identification of sitespecific artificial-recharge techniques in the Deccan Volcanic Province. Journal of Earth System Science, 114, 5, 505–514. Riad, Peter H.S, Max Billib, Ahmed A. Hassan, Maha Abdel Salam and Mohamed N. El Din, 2011. Application of The Overlay Weighted Model and Boolean Logic to Determine the Best Locations for Artificial Recharge of Groundwater. Journal of Urban and Environmental Engineering, 5, 2, 57-66. Saravi, M.M., Malekian, A., and Nouri, B., 2006. Identification of Suitable Sites for Groundwater Recharge. The 2nd International Conference on Water Resources & Arid Environment. 26–29 November 2006, King Saud University Riyadh. KSA. Saraf,A.K., P.R. Choudhury, 1998. Integrated remote sensing and GIS for groundwater exploration and identification of

artificial recharge sites. International Journal of Remote

Sensing, 10, 1825–1814.

Saaty, T.L., 1977. A scaling method for priorities in hier

archical structures. Journal of Mathematical Psychology,

15, 231–281.

Saaty, T.L., Vargas, L.G., 1991. Prediction, Projection and

Forecasting. KluwerAcademic Publishers, Dordrecht,The

Netherlands, 251 pp.

Schabenberger, O., Gotway, C. A., 2005. Statistical methods

for spatial data analysis. Chapman and Hall/CRC Press,

512.

Sharma, S.K., 1992. Remote Sensing techniques to locate

areas suitable for artificial recharge of groundwater in

granitic terrain Proc. Workshop Artificial recharge of groundwater in granitic terrain, Bangalore, 14-44.

Sirhan, H. and Koch, M., 2014. Numerical Prediction of

the Efficacy of treated Wastewater Recharge to impede

Saltwater Intrusion in the coastal Aquifer of GazaPales

tine, In: Proceedings of the 11th International Conference

on Hydroscience & Engineering, ICHE Hamburg 2014,

R. Lehfeldt and R. Kopmann (eds.), Hamburg, Germany,

September 28–October 2, 2014.

Sukumar.S, S., Sankar, K., 2010. Delineation of potential

zones for artificial recharge using GIS in Theni district, Tamilnadu, India. International journal of geomatics and geosciences, 1, 3, 639–648. Taheri, A., Zare, M., 2011, “Groundwater artificial recharge assessment in Kangavar Basin, a semi-arid region in the western part of Iran”, African Journal of Agricultural Research, 6, 17, 4370–4384. Vorhauer, C. F. and Hamlett, J.M., 1996. GIS: A Tool for Siting Small Farm Ponds, Journal of Soil and Water Conservation, 51, 5, 434–438. USDA, 1955. How much of the rain enters the soil? In Water: U.S. Department of Agriculture. Yearbook. Washington, DC., pp. 151–159. Zare, M. and Koch, M„ 2014. 3Dgroundwater flow modeling of the possible effects of the construction of an irrigation/drainage network on water-logging in the Miandarband plain, Iran. Basic Research Journal of Soil and Environmental Science, 2, 3, 29–39. Zehtabian, G. R., Alavipanah, S. K., and Hamedpanah, R., 2001. Determination of an appropriate area for flood water spreading by remote sensing data and GIS. In: Proceedings of the International Conference on New Technology for a New Century, Seoul, Korea, pp. 1–6. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Hydrological simulation in the Tiete basin

T. Rocha

School of Technology, State University of Campinas, Sao Paulo, Brazil

I.G. Hidalgo

Department of Energy System Planning, State University of Campinas, Sao Paulo, Brazil

A.F. Angelis

School of Technology, State University of Campinas, Sao Paulo, Brazil

J.E.G. Lopes

Department of Water Resources, State University of Campinas, Sao Paulo, Brazil

ABSTRACT: The operation planning of hydropower plants depends on the inflow forecasting. Usually, the inflow forecasting is done from mathematical, stochastic or hydrological models. This paper presents the use of Soil Moisture Model Accounting Procedure (SMAP) to predict daily inflows into hydropower plants. SMAP is a deterministic model of hydrological simulation of the type rainfall-runoff transformation. Our objective is to evaluate the accuracy of its results in order to support the decisions of water storage and power generation of hydropower plants. Generalized Reduced Gradient algorithm for nonlinear optimization is employed. SMAP performance is analyzed through three indicators: Nash-Sutcliffe efficiency, percent bias, and ratio of the root mean square error to the standard deviation. The results show that SMAP is a good alternative to predict water inflows. We hope to contribute for the improvement of the power system planning, avoiding the thermal complementation and reducing the electricity cost.

1 INTRODUCTION

Inflow forecasting is an important task especially for countries that have hydropower plants as main source of electricity. There are several models developed for predicting inflows. They can be conceptual (using, for example, hydrological simulation), empirical (using stochastic, statistic or neural networks techniques) or combined (associ ating the characteristics of the first two).

Conceptual models consider the physical processes of the water system to represent the variables.They can be classified as concentrated or distributed, accord ing to the space definition for the basin. MGB-IPH is an example of distributed conceptual model (Col lischonn & Tucci 2001).

Empirical models are based on the variables that describe the basin state. PREVIVAZ is an example of stochastic empirical model widely used in Brazil (Maceira et al. 1999). Statistic empirical model using linear regression can be found in Sousa Filho & Lall (2004). Luna et al. (2011) employ fuzzy infer ence systems in their empirical model for inflow forecasting.

Combined models associate the characteristics of

conceptual and empirical models. SMAP-MEL is an example of combined mode. It combines the SMAP (concentrated conceptual model) and MEL (stochastic empirical model). SMAP-MEL has been applied in the Parana River basin (ANEEL 2007). The inflow forecasting is a complex task important for the operation planning of hydropower plants. Despite the large number of models, there are still many forecast errors, as showed in Guilhon et al. (2007). Considering the importance of the energy sector for a country, the objective of this paper is to contribute with the analysis of the models' accuracy. The chosen model is Soil Moisture Model Accounting Procedure (SMAP) due its increasing use in Brazil. The model's performance is analyzed through three indicators recommended by Moriasi et al. (2007). They are Nash-Sutcliffe efficiency (NSE), percent bias (PBIAS), and ratio of the root mean square error to the standard deviation (RSR). NSE is a normalized statistic that determines the relative magnitude of the residual variance compared to the observed data variance (Nash & Sutcliffe 1970). PBIAS measures the average tendency of the predicted values to be larger or smaller than their observed ones (Yapo et al. 1996). RSR is calculated as the ratio of the root mean square error and standard deviation of the measured data (Moriasi et al. 2007).

Figure 1. Water cycle. http://www.eschooltoday.com/water cycle/the-water-cycle.html.

The objects of study are two hydropower plants of the Tiete River basin, in Sao Paulo state – Brazil. They are Ibitinga (IBI) and Bariri (BAR) started in 1969 and 1965, respectively.

The rainfall data are provided by the company that manages the hydropower plants, AES Tiete (AES, 2013). The historical inflows are obtained from a Brazilian agency named Centro de Previsão de Tempo e Estudos Climáticos (CPTEC 2013).

2 SOIL MOISTURE MODEL ACCOUNTING PROCEDURE (SMAP)

SMAP (Lopes et al. 1982, 2002) is a deterministic conceptual model of the type rainfall-runoff transfor mation which uses the water cycle concept, Figure 1. It considers the basin as a single system, so it is a concen trated conceptual model. In this paper its daily version is employed.

SMAP uses three estimated parameters based on physical processes.They are related to vegetation type, soil type, and flow rate in the study area. SMAP also employs three calculated parameters based on his torical series of precipitation and water inflow. They are capacity of soil saturation, constant of runoff recession, and parameter of groundwater recharge.

SMAP needs seven input variables: average precipi tation, evaporation rate, drainage area, initial moisture, initial basic inflow, initial runoff, and observed inflow in the day. The output variable is the predicted water inflow in the day.

SMAP consists of three water mathematical reser voirs. The reservoirs are of soil (R-soil), surface (R-surf) and underground (R-under). R-soil is the reservoir of the aerated zone. R-surf is the reservoir associated with the basin runoff. R-under simulates the reservoir of the saturated zone. Three equations and five transfer functions are used to calculate the reservoirs for each day. Details about SMAP can be found in Lopes et al. 1982. 3 METHODOLOGY The methodology consists of six steps that should be followed in order to prepare, apply, and evaluate SMAP model. They are: (1) analysis of the data consistency, (2) automatic calibration of the calculated parameters, (3) manual re-calibration of the estimated and calculated parameters, (4) adjustment of the measurement stations' weights, (5) validation/application of the model, and (6) evaluation of the results. First, the parameters and input variables should be analyzed, Step (1). Inconsistences, for example, in the water inflow data can be corrected using the linear interpolation technique. Then, the calculated parameters of the model are calibrated with Solver tool of Microsoft Excel, Step (2). According to Moriasi et al. (2007) a good calibration procedure uses multiple quantitative statistics. We use NSE, PBIAS, and RSR previously described. Third, the estimated and calculated parameters of the model can be manually re-calibrated in order to get better precision, Step (3). During the manual re-calibration, each quantitative statistic should be tracked for balancing the model ability and potential errors in the observed data (Boyle et al. 2000). In Step (4), the measurement stations' weights are adjusted also using Solver tool of Microsoft Excel. The algorithm employed by Solver in Steps (2) and (4) is the Generalized Reduced Gradient (GRG2) for optimizing nonlinear problems. This algorithm was developed by Leon Lasdon, of the University of Texas at Austin, and Allan Waren, of Cleveland State University (Excel 2015). In Step (5), the model is validated in a different period of the used for the calibration. After that, the model is ready to be used as a forecasting tool of water inflows. The results of the model are evaluated in Step (6). The predicted and observed water inflows are compared. For the calibration, adjustment, validation, and evaluation of the model we use graphics and statistics. Graphics provide a visual comparison of predicted and observed data and a first overview of the model's performance. Hydrographs, for example, can show differences in timing and magnitude of peak flows and the shape of recession curves. It can also show the model's tendency to underestimate or overestimate flow values on the whole horizon. Statistics are useful for a numeric analysis of the presented results. Table 1 shows the performance ratings for the indicators used in this paper, according to Moriasi et al. (2007). 4 CASE STUDIES AND RESULTS For the case studies the six steps of the methodology are carried out for IBI and BAR plants. Three study periods are defined for the calibration/adjustment, validation, and application of the model. SMAP is

Table 1. Performance ratings (Moriasi et al. 2007).

Classification	NSE	PBIAS (%)	RSR
Very good	$0.75 < NSE = 1.00$	$PBIAS < \pm 10$	$0.00 \leq RSR \leq 0.50$

Good 0.65 < NSE = 0.75 ±10 = PBIAS <±15 0.50 < RSR ≤ 0.60

Satisfatory 0.50 < NSE = 0.65 ±15 = PBIAS <±25 0.60 < RSR ≤ 0.70

Insatisfatory NSE = 0.50 PBIAS =±25 RSR > 0.70

Figure 2. Average daily precipitations for IBI.

Figure 3. Average daily precipitations for BAR.

calibrated from 2003 to 2005, validated from 2005

to 2007 and applied to January, April, and September

of 2010.

Figures 2 and 3 show the average daily precipita

tions for IBI and BAR, respectively. The data of these

figures correspond to three study periods of 2010;

1-15/Jan, 1-15/Apr, and 16-30/Sep.

Figures 4-6 present a comparison between pre

dicted (Q pred) and observed (Q obs) water inflow series

to three study periods of 2010; 1-15/Jan, 1-15/Apr,

and 16-30/Sep, respectively. They refer to IBI plant.

Table 2 shows, numerically, the same information.

Table 3 displays the values and ratings of the per

formance indicators for IBI; where (v) = very good,

(g) = good, (s) = satisfatory, and (i) = insatisfatory.

Overall, in the analyzed periods, 2/9 of the indicators

are classified as very good, 1/9 as good, and also 2/3

as satisfactory.

Figures 7-9 present a comparison between pre dicted (Q pred) and observed (Q obs) water inflow series Figure 4. Predicted and observed water inflows for IBI – Jan/2010. Figure 5. Predicted and observed water inflows for IBI – Apr/2010. Figure 6. Predicted and observed water inflows for IBI – Sep/2010.

Table 2. Predicted and observed water inflows for IBI.

	Jan/2010		Apr/2010		Sep/2010	
	Q pred	Q obs	Q pred	Q obs	Q pred	Q obs
Day	m^3/s	m^3/s	m^3/s	m^3/s	m^3/s	m^3/s
1	2400	2374	1220	1164	330	266
2	1968	1711	1111	1119	317	192
3	1670	1374	1076	1148	321	280
4	1602	1451	1092	1231	329	227
5	1584	1647	1267	1411	337	186
6	1577	1451	1075	1488	345	175
7	1571	1461	1023	1069	353	206
8	1564	1742	1001	1113	359	273
9	1612	1853	985	995	365	340
10	1572	1769	971	977	371	286
11	1554	1686	958	973	376	531
12	1547	1384	944	734	428	379
13	1542	1258	931	722	597	648
14	1536	1348	919	861	544	535
15	1532	1601	906	700	450	620

Table 3. Model's performance for IBI.

	Jan/2010		Apr/2010		Sep/2010	
Indicator	Value	Rating	Value	Rating	Value	Rating
NSE	0.53	s	0.52	s	0.52	s
PBIAS	-3	v	1	v	-13	g

RSR 0.68 s 0.69 s 0.69 s

Figure 7. Predicted and observed water inflows for BAR –

Jan/2010.

to three study periods of 2010; 1-15/Jan, 1-15/Apr,

and 16-30/Sep, respectively. They refer to BAR plant.

Table 4 shows, numerically, the same information.

Table 5 displays the values and ratings of the per

formance indicators for BAR; again (v) = very good,

(g) = good, (s) = satisfatory, and (i) = insatisfatory.

Overall, in the analyzed periods, 1/3 of the indicators

are classified as very good and 2/3 as satisfactory.

The results of IBI and BAR for 1-15/Jan/2010 using

SMAP are compared with the results from a model Figure 8. Predicted and observed water inflows for BAR – Apr/2010. Figure 9. Predicted and observed water inflows for BAR – Sep/2010. Table 4. Predicted and observed water inflows for BAR Jan/2010 Apr/2010 Sep/2010 Q pred Q obs Q pred Q obs Q pred Q obs Day m 3 /s m 3 /s m 3 /s m 3 /s m 3 /s m 3 /s 1 1850 2045 1040 1023 250 174 2 1503 1442 956 923 203 196 3 1333 1168 928 956 199 229 4 1292 1130 940 1027 203 154 5 1278 1239 1067 1355 209 159 6 1270 1184 927 1162 215 146 7 1263 1246 883 1071 221 161 8 1257 1511 864 856 226 203 9 1273 1619 851 882 231 239 10 1254 1501 839 797 235 269 11 1240 1399 827 788 239 382 12 1232 1220 816 589 319 335 13 1225 1042 804 593 473 497 14 1218 1088 793 661 381 407 15 1220 1359 782 570 298 501 based on Fuzzy Inference Systems (FIS). The technique used for adjusting the parameters of the FIS model is an offline version of the Expectation Maximization algorithm. Details about the FIS model can be found in Luna et al. (2011).

Table 5. Model's performance for BAR. Jan/2010 Apr/2010 Sep/2010

Indicator Value Rating Value Rating Value Rating

NSE 0.52 s 0.52 s 0.60 s

PBIAS 2.4 v -1 v 4 v

RSR 0.69 s 0.69 s 0.63 s

Figure 10. Predicted and observed water inflows for IBI –

Jan/2010, using the FIS model.

Figure 11. Predicted and observed water inflows for BAR –

Jan/2010, using the FIS model.

Figures 10-11 present a comparison between pre

dicted (Q pred) and observed (Q obs) water inflow series

for IBI and BAR, respectively; using FIS model.

They refer to 1-15/Jan/2010 period. Table 6 shows,

numerically, the same information.

Table 7 presents the percentage relative deviations

for IBI and BAR in the three study periods of 2010;

1-15/Jan, 1-15/Apr, and 16-30/Sep, using SMAP

model. For 1-15/Jan/2010, using the FIS model, the

percentage relative deviations for IBI and BAR are

30% and 33%, respectively.

5 SUMMARY AND CONCLUSIONS

This paper presents an application of SMAP hydro

logic model to predict daily water inflows for two Table 6. Predicted and observed water inflows for IBI and BAR, using the FIS model. IBI BAR Q pred Q obs Q pred Q obs Day m 3 /s m 3 /s m 3 /s m 3 /s 1 1100 2374 1100 2045 2 1245 1711 1168 1442 3 1895 1374 1653 1168 4 1878 1451 1403 1130 5 1528 1647 1071 1239 6 1211 1451 839 1184 7 1146 1461 842 1246 8 1087 1742 898 1511 9 1094 1853 913 1619 10 1112 1769 885 1501 11 1110 1686 846 1399 12 1089 1384 822 1220 13 1057 1258 805 1042 14 1008 1348 787 1088 15 958 1601 799 1359 Table 7. Percentage Relative Deviations for IBI and BAR. Plant Jan/2010 Apr/2010 Sep/2010 IBI 11% 11% 36% BAR 11%

15% 22% Brazilian hydropower plants, IBI and BAR. The objective is to contribute with the accuracy of an important input variable for the hydropower system planning. The steps to prepare, apply, and evaluate the results of SMAP model are described. Two case studies, with three periods each one, are shown. The results are analyzed through of the indicators NSE, PBIAS, and RSR; recommended in the literature. The indicators confirm a good performance of the SMAP model with 100% of the results classified into “very good, good, and satisfactory” range, in a distributed way. The percentage relative deviations for IBI and BAR using SMAP model are, on average, 19% and 16%, respectively. These values are good comparing to the FIS model and the annual reports of inflow forecasting for Tiete River basin that show a deviation between predicted and observed water inflows up to 30%. It is important to mention that in the presented case studies the model deal with potential errors in the data, but not with uncertainties in the data because the employed precipitation data are observed. However, the ranges of the performance indicators (Table 1) were established for a monthly time step. Therefore, they are stricter than necessary. As future work the authors intend to broaden the case studies with SMAP model. It can be applied for more plants or measurement stations in the Tiete River basin and for different periods, including dry and wet seasons.

AES. 2013. AES Tiete company. Available at: http://www.

aestiete.com.br/.

ANEEL. 2007. New streamflow forecasting model with rain

fall information for the incremental portion of Itaipu (in

Portuguese). Available at: http://www.aneel.gov.br.

Boyle, D. P.; Gupta, H. V.; Sorooshian. S. 2000. Toward

improved calibration of hydrologic models: combining

the strengths of manual and automatic methods. Water

Resources Research, 36(12): 3663-3674.

Collischonn, W. & Tucci, C. E. M. 2001. Hydrologic simula

tion of large basins (in Portuguese). Revista Brasileira de

Recursos Hídricos, 6(1): 95-118.

CPTEC. 2013. Centro de Previsao de Tempo e Estudos Cli

maticos (Brazilian agency).Available at: www.cptec.inpe.br.

Excel. 2015. Available at: https://support.microsoft.com/en

us/kb/82890.

Guilhon, L. G. F.; Rocha, V. F.; Moreira, J. C. 2007. Compar

ison of natural inflows forecasting methods to hydroelec

tric plants (in Portuguese). Revista Brasileira de Recursos

Hídricos, 12(3): 13–20.

Lopes, J. E. G.; Braga B. P. F.; Conejo J. G. L. 1982. SMAP
–

A simpliflied hydrologic model.Water Resourses Publica

tion (Org.). Applied Modeling in Catchment Hydrology.:

Hittleton 167–176. Lopes, J. E. G.; Braga B. P. F.; Conejo J. G. L. 2002. A simple hydrological model, applied modelling in catchment hydrology. Ed. V. P. Singh: Water Resources Publications. Luna Huamani, I. R.; Hidalgo, I. G.; Barbosa, P. S. F., Francato, A. L. 2011. Daily reservoir inflow forecasting using fuzzy inference systems. IEEE International Conference on Fuzzy Systems, Taipei, Taiwan: 2745–2751. Maceira, M. E. P.; Damazio, J. M.; Ghirardi, A. O.; Dantas, H. M. 1999. Periodic ARMA models applied to weekly streamflow forecasts. International Conference on Electric Power Engineering. Budapest – Hungary. Moriasi, D.; Arnold, J. G.; Van Liew, M. W.; Bingner, R.; Harmel, R.;Veith,T. L. 2007. Model evaluation guidelines for systematic quantification of accuracy in watershed simulations. Transactions of the ASABE, 50(3): 885–900. Nash, J. E. & J. V. Sutcliffe. 1970. River flow forecasting through conceptual models part I – A discussion of principles. Journal of Hydrology, 10(3): 282–290. Souza Filho, F. A. & Lall, U. 2004. Prediction model of seasonal and interanual inflows (in Portuguese). Revista Brasileira de Recursos Hídricos, 9(2): 61–74. Yapo P. O.; Gupta H. V.; Sorooshian S. 1996. Automatic calibration of conceptual rainfall-runoff models: sensitivity to calibration data. Journal of Hydrology. 181(1-4): 23–48. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN

978-1-138-02977-4

Hydraulic analysis of an irrigation headworks complex

in the Artibonite department (Haïti)

S. Louis & N. Gonomy

Département de Génie Rural, Faculté d'Agronomie et de Médecine Vétérinaire (FAMV),

Port-au-Prince, Université d'Etat d'Haïti (UEH)

N. de Ville & M.A. Verbanck

Department of Water Pollution Control (CP208), Université Libre de Bruxelles (ULB), Belgium

ABSTRACT: In the Northern part of the Republic of Haiti, water is extracted from the Artibonite River to

irrigate 300 km^2 of a wide agricultural plain constituted primarily of paddy fields. The river comes from the

Peligre reservoir where it is impounded (altitude 150 m, close to the border with Dominican Republic) to reach

the Canneau irrigation dam (altitude 32 m) barring its course 70 km further downstream. Due to the variable

Peligre releases and the rains which can solicit hydrologically the tributary catchments located between the two

dams, the river level at Canneau is rarely keeping stable. The purpose of the Canneau dam is to facilitate the

operation of two main irrigation channels, one which runs parallel to the river on the left side and a second one

on the right side. The Left-side Master Canal (CMRG in its French denomination) is at the center of our analysis

and investigative efforts. The water intake at Canneau is controlled by the opening of a cable-hoisted radial gate

(vanne-segment) fitted with counterweight. The nominal design discharge sent to the CMRG is intended to be

40 m^3/s. A first objective of the local experimental work

is to calibrate the flow rate at the gate with the adequate combination of valve opening heights (creating either free or submerged orifice conditions) and the explicit account of temporal changes in the headwater tail. A second objective is to develop a 1D velocity model valid for the first two kilometers of the trapezoidal earth-lined CMRG canal. It is shown that there is a specific stream power range in the CMRG for which the alluvial resistance is nicely described by the integer value of control factor m = 3 (Verbanck 2008).

1 INTRODUCTION

1.1 General setting

As part of an inter-university cooperation programme conducted between French-speaking Belgium and the Republic of Haiti, a ULB doctoral engineering project is presently underway in the Caribbean's. Its general objective is to assess and investigate the water and sed iment budget at a key hydraulic site in the Artibonite Department, namely an irrigation headworks complex (see Section 1.2) which is located 143 km to the North of the Haitian capital Port-au-Prince. The Artibonite River forms the largest surface-freshwater resource in the country (it is also one of the largest in Hispaniola Island). It drains a transboundary watershed totalling 9000 km^2 , one third of which, to the East, lays in the neighboring Dominican Republic. Close to the national border, the water regime of the Artibonite is under the strong and determining influence of a major reservoir (Peligre Dam, an artificial lake impounded in 1956) which drains 6480 km^2 of the upper watershed and which had, at the time of construction, a nominal volume capacity of 470 million m^3 (Morris 2009). However, already in 1982, the seriousness of sediment deposition problems affecting Peligre waterstorage (about one half of which is presently lost due to sedimentation) became apparent together with the downstream riverbed-degradation risks induced by the sedimentological imbalance it creates throughout the Artibonite valley over its course towards the sea (Frenette et al. 1982). Besides its role in contributing to the alleviation of flood risks in the Artibonite valley and the production of electricity through its turbines (3 × 17 MW), the steady water releases by Peligre Dam are obviously very beneficial in supporting

Figure 1. Canneau dam (upstream view).

irrigation needs at times of severely low waterflows (see below).

1.2 The irrigation headworks complex at the diversion site Canneau

The part of the Artibonite valley considered in this study is the area between (18∘50′ and 19∘18′ N) of latitude and (71∘37′ and 72∘47′ W) of longitude. Seventy kilometers downstream Peligre reservoir there is, on the river, an important irrigation dam at Canneau, close to the municipality of Petite-Rivière de l’Artibonite. The main purpose of the Canneau dam is to maintain, during periods of normal riverflow, a constant water surface elevation and, by doing so, to provide satisfactory water diversion volumes to the local

(Master) irrigation canal intakes. Except during peri ods of high river discharge, this forebay elevation at the Canneau diversion dam is normally maintained (if Peligre releases actually permit) at 29.8 meters above sea level.

Note that, during part of our monitoring days for the 2015–2016 survey, this 29.8 m head could at times only be attained through a complete shutting down of the riverflow-transfer locks fitting the Canneau dam. This demonstrates the acuteness of the drought cri sis presently threatening the country's economy and severely affecting its agricultural rice growth pro duction (for years, the paddy fields of the irrigated Artibonite plain have constituted the major food supply to the Haitian population).

Close to the left bank of the Artibonite River and a few meters upstream Canneau dam, there are two water intakes corresponding to a total (nominal) discharge of 50 m^3/s. This ensures feeding through a gravity sys tem, of a wide irrigated rice-growing plain totaling about 32,000 hectares of rice land.

The Master irrigation canals are laid at a 1% topo graphical slope, which is smaller than the bed slope of the Artibonite downstream Canneau, meaning that water which would be present in excess in a Mas ter canal (through inappropriate gate opening, for instance) can be returned to the river through lateral weir-flow simply driven by gravity. Such an overflow structure (see Section 1.3) and its possible role in the overall water budget is part of the investigative work presently underway.

1.3 The left-bank Master Canal

The earth-lined Left-side Master Canal (CMRG in its French denomination) is at the center of our analysis and investigative efforts. The water intake at Can neau is controlled by the opening of a cable-hoisted radial gate (vanne-segment) fitted with counterweight (Fig. 2). The width of the gate is 6.25 m and its open ing, from 0 ′ to 6 ′ (six inches), primarily drives the amount of irrigation water which is diverted from the river. Figure 2. CMRG Radial gate. 1.4 General problems to be addressed The nominal design discharge sent to the CMRG is intended to be 40 m^3/s. In case this value being exceeded it is anticipated, again by design, that the overflow structure located at first iron-footbridge, which is 2.25 km downstream the dam, will start operation and return the excess water back to the river, which is there very close by (Fig. 3 & 4). We are going to check this through detailed discharge observations (see Section 2 regarding the implemented methods). The aim is to check this 40 m^3/s pivotal value, firstly in terms of what is delivered at the radial gate for a given angular opening and, secondly, in terms of what is effectively remaining (or not) of the 40 m^3/s at the ironfootbridge check structure (Fig. 3). It’s very important to know the flow at Iron Bridge, because it is the actual flow that irrigates the 24,300 ha of rice land downstream and the feeding of the Drouet hydroelectric power plant (2500 kW capacity). The Canneau gatemen have large experience in achieving this 40 m^3/s delivery value but, surprisingly enough, they have relatively low consideration for the actual river head on the upstream side of the Canneau dam. Moreover, we want to identify the full hydraulic response (in its present stage before retrofitting, instrumentation and automatization (Milési et al. 2013) of the headworks radial gate. The other specific questions we want to address along the same lines are the following. What is the 1D velocity of the flow transiting in the CMRG for all possible valve openings? What are the stream velocity and effective shear stress conditions, more particularly at the iron bridge check structure, when the overflow starts functioning? Obtaining this set of key information is especially important if we also want to protect the canal side banks and avoid wastage of water, while protecting downstream population. This work therefore also conducts a thorough hydraulic analysis of the CMRG canal in its first 2.25 km-long reach, starting from Canneau dam and ending at the iron-bridge (Fig. 3), where a full discharge check structure has now been installed. Whenever possible, observed water-surface gradients will be part of the analysis.

Figure 3. Iron-bridge and overflow structure.

1.5 The specific objectives of this part of the doctoral study

Considering the above issues, we have set the follow

ing objectives:

– Estimate the flow rate into the CMRG by the (usu

ally submerged) orifice flow equation (Collins et

al 1977).To achieve this we must experimentally

determine the value of μ coefficient in the flow

equation through an orifice.

– Determine the flow at Iron-Bridge Overflow, from

the average velocity in the CMRG that will be

approached in two ways: a) Develop a 1D veloc

ity model without using a rating curve, since it is

not possible to establish a unique stage-discharge relationship for the measurement test site; b) Set a surface velocity conversion coefficient (Us) to average velocity (U¯) that is representative of the local flow conditions.

2 DESCRIPTION OF EXPERIMENTAL

The tools, methods and instruments to reach the goals presented above are numerous. The study focuses on the first 2.25 kilometers of the Master Canal described above, constituting the natural laboratory of experi mentation.The test measurement device set up consists of several measurement stations:

- A station upstream of the dam Canneau or CMRG sector valve to reflect the upstream head (I);
- A Vegapulse-operated limnimetric station down stream (J) of the radial valve to appreciate the height of its tail waterbed;
- Three staff gauges downstream for the observations of water-surface slopes (A, B and C);
- The iron bridge with overflow (Fig. 3) is the reference flow measuring station.

The wetted area versus height relationship is estab lished there. It remains to establish the 1D flow velocity (U¯) to this place to get the flow, including
Figure 4. Presentation of the experimental measuring set-up in the first 2.25 km of the CMRG. Table 1. Elevation of installed stations. Station Elevation (m) Distance from Dam (m) Dam (BM) 32.004 0 Upstream (J) 20.144 20 A 25.447 282 B

25.260 1307 C 25.362 2000 Iron-Bridge 24.070 2132 at conditions and exact time of starting of operation of the overflow structure (Fig. 3). The following table provides the absolute elevations of the limnimetric stations installed. 3 BASIC CONCEPT OF THE METHOD To achieve the specific objectives set previously, we have adopted the following methodology. 3.1 Calculation of the incoming flow of the CMRG using the submerged-orifice law To determine how much water gets into the CMRG under varying operating conditions, we turn to the submerged-orifice law (Equation 1) in which we have to tune the flow coefficient μ (Collins 1977). The approach requires simultaneous experimental information on three quantities: – the flow rate (Q) obtained 30 m downstream of the valve using a portable electromagnetic currentmeter OTT (MF Pro); – The upstream head (h_1) with a staff gauge; – The tail waterbed (h_3) using the radar limnimeter and long temporal integration (resolution 30 s); The opening height of the radial gate (h_g) is also recorded. where Q = discharge (m^3/s); μ= submerged-orifice flow coefficient of discharge (–); h_1 = staticheadwater depth (m); h_3 = static-tailwater depth (m);

B = radial gate width (m); g = gravity acceleration

(m/s^2).

3.2 Determination of the discharge at Iron-Bridge overflow

The discharge at Iron-Bridge has been determined by

the following relation

where U = average velocity (m/s); A = wetted

area (m^2).

A water depth – wetted cross section relationship

is established in the control section (reference). It

remains to locally determine the cross-section aver

aged velocity.

The measurement of the 1D average velocity of a

flow requires a set of equipment not accessible eas

ily, especially in CMRG where the average width is approximately 20 m and the average depth is of 3 m.

For this, two methods were used to assess the average

velocity: use of a 1D velocity model and use of the

surface velocity measured in the field with a velocity

coefficient.

3.2.1 Use of a 1D velocity model to determine the average velocity of the flow

To determine the average velocity of the flow, we

rely on the vortex-drag model proposed by Verbanck

(2008). This is a 1D velocity model relying on system

atic water-slope observations per-formed all along the

CMRG length. It takes into account the control fac

tor m concept. The latter, which should in theory be

locked at integer values (m = 1, m = 2, m = 3 . . .) is a

quantity related to the morphological adaptation of the

bed and the ensuing flow resistance. It is considered

the only parameter that would control the interactions

of alluvial rivers. The average flow velocity from the

model ($\bar{U}_m$) is given by the following equation:

where $\bar{U}_m$ = 1D velocity (m/s); m = control factor

(–); S = slope of the energy grade line (–);

R = hydraulic radius (m).

The model proposed is partly based on the univer

sal law of Strouhal developed by Levi (1983). This

law expresses the fact that the frequency of distur bances created by a fluid flowing around an obstacle

is defined only by the fluid velocity and a characteris

tic length scale of the considered system. In this case

the perturbations are created by the bedforms and this

is manifested as kolk-boil vortices which become reg

ularly visible at the water surface. Measurement of

frequency of occurrence of vortex in the surface of

the water has already been performed on the Parapetí

River in Bolivia (deVille et al. 2013) with encouraging

results.

As part of CMRG analysis, it was possible to deter

mine a complete and unique value of control factor m in a specific stream power range (ω). The specific stream power expresses the power of the current per unit area of bed in W/m 2 , according to Bagnold (1966). where ρ= density of water (kg/m 3); h = water depth (m). Experimental verification of the integer value m In the CMRG, the local observations of bathymetry along the longitudinal axis indeed show the presence of dunes (or mini-dunes) with approximately 3 m wavelength (λ BF). Thus, to verify the integer value m, we have used the Strouhal law defined below: where T = boil period (s); m = control factor (-); x r = characteristic length (m). In the case of fine-gravel, the ratio between x r and λ BF was taken as 0.82 (Verbanck 2008). 3.2.2 Use of the surface velocity measured in the field with a velocity coefficient to determine the average velocity of the flow The surface velocity is measured in the field using floats. Now, we try to identify the value of the alpha velocity conversion coefficient for converting the surface velocity, easily measured in a repre-sentative average flow velocity considered accord-ing U¯ =αUs equation. To achieve this, two complementary approaches are used: a) Conceptual approach by combining the log-arithmic velocity profile with the 1D vortex-drag velocity model presented in §3.2.1; b) experimental approach by exploration of the velocity field. a) Conceptual approach by using a 1D velocity model and logarithmic velocity profile First, we have determined the roughness Reynolds number in the CMRG by the following equation: where Re * =

local Reynolds Number (-); d_{65} = grain diameter (m) (Table 6); u_* = friction velocity (m/s); v = kinematic viscosity of water (m^2/s) at 30 ∘ C. We have obtained a roughness Reynolds number greatly exceeding 70. Thus, we can consider that we are in rough boundary flow regime in the CMRG. In that case, the flow velocity in the inner flow region ($z/h < 0.20$) at the bottom follows a logarithmic profile which simply depends on the value of roughness coefficient k_s (Equation 9 and Equation 12). Once integrated over the depth it gives, Keulegan (1938):

where k =Von Kárman constant ($k = 0.4$); R =

hydraulic radius (m); k_s = flow roughness (m);

u_* = friction velocity (m/s).

Determination of equivalent roughness

In the CMRG near the control section, we observed the

presence of fine-gravel dunes and kolk-boils. In such a

case, the k_s value is dominated by bedform roughness

rather than particle size. Thus, the k_s value has been

deducted from the equation 9, with $\bar{U} = \bar{U}-m$ and the

control factor imposed as $m = 3$.

We also remember that a bedform roughness can be

approximated, Swart (1976) as

where k_s = hydraulic bed roughness (m); r = bed

form height (m); λBF = bedform length (m); r is

approximated according to Flemming (2000) rule:

Once the roughness k_s is obtained, the $u(z)$ profile can

be expressed by equation 12.

where $u(z)$ = local velocity (m/s); z = water depth (m).

Dividing the average velocity (equation 3) by the

surface velocity (upper part of equation 12) for a given set of measurements, a predicted value of the alpha coefficient of velocity can be obtained.

b) Average flow velocity obtained with the method of exploring the vertical velocity field

The exploration of the velocity field using a portable electromagnetic current meter (OTT MF Pro) allowed to study the vertical velocity distribution in the reference section (iron-bridge check structure).

The number and the location of points in the vertical have been determined by the approaches of Rantz (1982) and of Boiten (2002). But, the determination of the mean velocity in the vertical has been done according to Rantz (1982).This method was deployed for various gate openings. During this deployment, surface velocities using floats were also measured to obtain an

experimental value of the reference coefficient α. Table 2. Discharge coefficient. Q (m 3 /s) H 1 (m) H 2 (m) ΔH (m) μ (-) 37.45 28.94 26.83 2.11 0.61 4.825 28.50 25.77 2.73 0.69 5.12 28.54 26.03 2.51 0.65 Figure 5. Comparison of the velocity profile for three successive flow events (Iron-bridge overflow). 4 RESULTS AND DISCUSSION 4.1 Flow coefficient of the radial gate from submerged-orifice law Only a few observations could be collected to calculate the flow coefficient through the 6.25 m width valve. The following table shows the coefficients obtained from the measurements made. Compared to the values presented in the literature by Collins (1977) and Alexander (1992), the flow coefficient μ= 0.62 seems reasonable. This value, when it will be confirmed, could be used to calculate throughput per orifice embedded in the CMRG. 4.2 Mean velocity and alpha experimental using the method of exploration of the velocity field Several velocity profiles (Table 3) have been measured in the water column on three different days with very different flow rate conditions. According to

Prony (Graf 1998), the surface velocity coefficient varies from 0.80 to 0.90. But, in a natural channel a surface velocity coefficient of 0.85 or 0.86 is used to compute mean velocity, Rantz (1982). As can be seen the value of the alpha coefficient changes significantly when the flow conditions are very different. When water levels are important (3 m during measurement of 4 February 2016) a flow area (+-0.6 y/D) undergoes less resistance to flow and flows faster than the velocity of surface, which has the effect of increasing the value of the alpha coefficient. Subsequent research efforts will be needed to validate these results and possibly determine whether an unambiguous relationship between the water level and the speed factor may well be established.

Table 3. Velocity profile.

Date and hour	h (m)	U (m/s)	U_s (m/s)	α
01 /15/2016	1.55	0.343	0.245	0.81
02/04/2016	2.91	0.810	0.870	0.93
23/04/16:8:00	2.30	0.433	0.54	0.80
23/04/16:10:00	2.32	0.690	0.73	0.95
23/04/16:11:50	2.49	0.709	0.79	0.90
23/04/16:12:50	2.68	0.729	0.85	0.86

Figure 6. Influence of specific stream power on control

factor m values.

4.3 Mean velocity using the validated 1D velocity model

Control factor m

For handling the 1D velocity model and calculating

the roughness coefficient k_s , it is necessary to fix a

value of control factor m. In the specific stream power

range of CMRG observed according to the chart below,

we see that the measured control factor systematically

takes a value around 3. So we can afford to make the assumption of a constant & integer value for control factor m = 3 for specific stream powers sufficiently close to 5 W/m 2 .

From a practical and application point of view, this is a very interesting result, because of its simplicity in the flow resistance calculations.

Moreover, from a conceptual point of view, we present below a tentative experimental verification of this integer value m = 3, using the method of extract ing the periods of the successive kolk-boils raised at the back of the riverbed dunes, and which appear at the water surface at recurrent intervals. At this stage of our work, the value m = 3 is imposed in Equation 2 and the average velocities are calculated using the measured water-surface slopes and water heights. The expected frequency for a control factor m = 3 is cal culated with Equation 5 (Strouhal law) and the results are presented in Table 5.

As can be seen, the measured period is very close to the calculated period. This is encouraging for the implementation of the 1D velocity model. For a con trol factor m = 3, real measured water levels and wa ter slope of observations, one can calculate average velocities. Table 5. Observed periods of kolk boils. Parameter Value m 3 U (m/s) 0.93 X r (m) 2.42 Tcomputed (s) 5.4 Tmeasured (s) 6 Figure 7. Velocities model and measured in April 2016. 4.4 Comparison of the two average velocities (model and measured) obtained For the same height of water flow conditions observed inApril 2016, we have compared the average velocities obtained by the 1D velocity model (U_m) with average velocities obtained by the method of exploring the velocity field (U_C). For these observations, Figure 7 shows that the modeled velocity is very close to the vertically-integrated OTT velocity measured for different valve openings (1.5 ′ , 2.5 ′ , 3.5 ′ and 4 ′). 4.5 Determination of CMRG flow at Iron-Bridge overflow The velocity being known, Equation 2 can be used to evaluate the CMRG flow rate at the Iron-Bridge. It allows to build a diagram with three entries (Fig. 7): flow rate, opening of the valve and the upstream head load. The upstream head, in relative terms, was cut into a set of intervals. This graph shows that the flow rate is a function of the gate opening of the valve, but also of the upstream head. For the same valve opening, one gets multiple flow rates corresponding to different upstream head loads. According to this graph, the regular opening of the valve is 5 ′ . Moreover, the flow rate corresponding to the start of operation of the overflow to the gateway (at least under the usual conditions of hydraulic gradients in the CMRG) is 35 m^3/s. 4.6 Determination of equivalent roughness By sieving of deposits sample taken in the reference section, we have obtained the following grain size characteristics as in Table 6.

Figure 8. Abacus giving the flow rate according to vertical

height of radial gate opening.

Table 6. Grain size of deposits in the reference section.

D_{10} (mm)	D_{50} (mm)	D_{65} (mm)	D_{90} (mm)
0.39	3.73	7.21	15.78

Table 7. Channel roughness obtained from first profile in

April 2016.

y/h	H (m)	k_s (m)
0.05	0.115	0.262
0.2	0.46	0.258

From the boils observed in the CMRG and the meth ods of Flemming (2000) and of Swart (1976), we have

obtained a bed roughness coefficient (k s = 0.224 m).

We use the logarithmic universal law, in the lowest

20% of the flow depth (Nezu & Rodi 1986) to extract

the value of the effective roughness from the verti

cal profiles of April 2016. The roughness coefficient

obtained by the boils observation technique is close to

the k s obtained on the 20% in the lower part of the ver

tical profile.The roughness coefficient obtained by the

boils observation technique is close to the k s obtained

on the 20% in the lower part of the vertical profile.

If we compare this result with that obtained by siev

ing, we can say that the bed-form dominates CMRG

roughness. This permits us to consider that our bed

form roughness is validated, and should normally help

us in our study of the alpha coefficient (§4.7).

4.7 Velocity coefficient alpha obtained by using the logarithmic universal law and the control parameter m

For each water level measured, the local velocity was

generated (Equation 12) cm per cm. For observations

in April, the results are presented in Table 8.

These surface velocities computed (Us_cal) are

compared with surface velocities measured (U_obs) Table 8. Alpha coefficient by the conceptual approach. h U_m u * k s U_cal Us_cal (m) (m/s) (m/s) (m) (m/s) (m/s) α 2.3 0.56 0.05 0.22 0.61 0.72 0.84 2.42 0.71 0.07 0.22 0.76 0.93 0.82 2.59 0.72 0.07 0.22 0.77 0.93 0.83 2.68 0.78 0.07 0.22 0.832 1.01 0.82 Figure 9. Evolution of Us_cal and Us_obs relative to water level. in the field. As shows the Figure

9 below, the logarithmic universal law overestimates the surface velocity. Accordingly, the value of alpha decreases. Therefore, the law evaluates more or less the average velocity. Furthermore, dividing the average velocity (equation 9) by the surface velocity (equation 11) for each measurement, a deduced value of the alpha coefficient of velocity is obtained. This value varies from 0.85 to 0.86. These values should be taken with caution. According to Figure 5, the velocity profiles in iron-bridge do not follow a logarithmic law (except in the 20% from bottom). Combination of a wake effect and secondary currents laterally makes that, most of the time, the experimental flow velocity close to the surface is significantly smaller than predicted by the Log-law (although the bedform roughness was adequately tuned). Accordingly an enhanced Log-Wake law (Sarma et al. 2000) is presently under development for the specific case of the CMRG. 5 CONCLUSIONS The experimental set-up of measurement that is in place has permitted to collect hydraulic data such as surface velocities, water depth and water-surface slopes. Examined in depth, the first data collected have permitted to determine incoming flow rate into the CMRG for different gate openings (0–6 ′), and the flow in the Iron-Bridge, located 2 km below the dam. The incoming flow rate into the CMRG has been studied in depth by the submerged-orifice law and a flow coefficient (μ) set at 0.62. Knowing all these parameters (inputs), it is now possible to determine the

incoming flow rate into the CMRG via this diversion

structure.

The flow rates obtained at the iron-bridge enabled

the start of construction of an abacus. It shows that

operating the beginning of the overflow to a rate of

35 m^3/s corresponds to a gate opening of 5 ′ , at least

when the water slope presents its usual values.

A α-velocity conversion coefficient was set at 0.80.

A 1D model velocity, based on water-surface slope

observations is now validated by a control factor

($m = 3$) in the specific stream power (close to 5 W/m^2)

of CMRG. The effective k s value obtained by two independent methods could be validated for the inner flow region. Further research, however, should exam ine deviations from the Log-Law in the outer region of the flow, notably due to the relatively narrow channel geometry (W/h around 5.7).

Bagnold, R.A. 1966. An approach to the sediment trans port problem from general physics, US Geological Survey Professional Paper 422-I, U.S. Department of the Interior.

Boiten, W. 2002. Flow measurement structures. Flow Mea surement and Instrumentation 13(5): 78–87.

Collins, D.L. 1977. Computation of Records of Stream flow at Control Structure U.S. Geological Survey, Water Resources Investigation Report 77-8.

de Ville, N. & Petrovic, D. & Verbanck, M.A. 2013. On the role of fine-sand dune dynamics in controlling water depth changes in Rio Parapeti, Serrania Borebigua (Southern sub-Andean zone of Bolivia), In: Marine and River Dune Dynamics (edited by V. Van Lancker & T. Garlan), VLIZ

Spec Publ 65, 73–80. Flemming, B.W. 2000.March. The role of grain size, water depth and flow velocity as scaling factors con-trolling the size of subaqueous dunes. In Marine Sandwave Dynamics, Intern. Workshop (pp. 23–24). Frenette, M., Tournier J.P. & Nzakimuena T.J. 1982b. Cas Historique de Sédimentation du Barrage Péligre, Haïti. Canadian Jour. Civil Engineering 9(2): 206–223. Graf,W. &Altinakar, M.S. 1998. Fluvial Hydraulics Flow and Transport Processes in Channels of Simple Geometry". Keulegan, G. H. 1938. Laws of turbulent flow in open channels: U.S. Natl. Bur. Standards, Jour. Research 21(6):

707–741. Levi, E. 1983. A Universal Strouhal Law. J. Eng. Mech. 109: 718–727. Milési, G. & Riffard, M. & Rodet, J. & Favreau. G. 2013. Etudes d'instrumentation des barrages de Péligre et de Canneau-Besoin en investissement. Tractebel engineering, Coyney et Bellier, 73. Morris, G. L. Engineering. 2009. Sedimentation Study of Peligre Reservoir, Haiti, Inter-American Development Bank, Washington, DC. Nezu, I. & Rodi, W. 1986. Open-channel flow measurements with a laser Doppler anemometer. Journal of the hydraulics Division 112, 335–355. Rantz, S.E. 1982. Measurement and Computation of Stream flow: Volume 2. Computation of Discharge. U.S. Geol. Surv. Water Supply Pap. 2175. Sarma, K. V.N. & B. V. Ramana & Sarma A.K. 2000. Detailed study of binary law for open channels, J. Hydraulic Engineering 126(3): 210–214. Swart, D.W. 1976. Predictive Equations Regarding Coastal Transports. In Coastal Engineering (pp. 1113–1132). ASCE. Verbanck, M.A. 2008. How fast can a river flow over alluvium. Journal of Hydraulic Research 46(1): 61–71. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Effect of emergent vegetation distribution on energy loss

E. Eris¸, G. Bombar & Ü. Kavaklı

Ege University, I˙zmir, Turkey

ABSTRACT: For the ecological aspect, it is well known that vegetation has fundamental functions in the river

environment. For fluvial hydraulics, on the other hand, presence of vegetation could be considered as a problem

for channel flow capacity due to its roughness. In order to understand the effects of planform areal configuration of

flexible vegetation on flow, a set of experiments were performed in the Hydraulics Laboratory of the Department

of Civil Engineering, Ege University, in a rectangular flume which is 5 m long, 18 cm wide and 20 cm deep

with transparent sides. Experiments were conducted with two different spatial distributions of vegetation, single

side (grasses positioned on one half of the flume) and double sided and preserving the total number of grasses

same. For each distribution type different downstream flow

depths and different discharges were applied and in total 75 runs were performed with different Froude numbers. By conducting experiments without any vegetation the average base head loss for the test section was obtained. Energy losses were calculated using the energy balance and the continuity equations for all scenarios both with and without vegetated conditions. The results were compared and interpreted. Friction factors for each scenario were determined.

1 INTRODUCTION

Study of river behavior and effect of vegetation on river systems are very important since the velocity, flow depth, suspended sediment transport and bed load are strongly influenced by the presence of vegetation (Vargas-Luna et al. 2015a&b).The vegetation induced drag reduces the flow discharge in channels increase the hydraulic roughness and helps flood attenuation and sediment deposition (Cheng & Nguyen 2011). Resistance due to vegetation varies in vegetation zones of river and flood plains due to vegetation flexibil ity, rigidity, submergence, foliage and side-branching, type, height, density and spatial distribution of plants (Fisher 1996, Fathi-Moghadam & Drikvandi 2012) (Fig. 1).

Predicting the vegetation resistance is very com plex since the combination of the different species with different characteristics changes during the sea son as well. Most vegetation types have foliage and side-branches making the vegetation resistance more complicated. In some researches the effect of leafs on the roughness was investigated. Jarvela (2002) revealed that the drag coefficient for leafy willows was about three to seven times more than leafless wil lows. According to Wilson (2007) and Freeman et al. (2000), the flow resistance of a plant may be signifi cantly less for a flexible plant with considerable foliage compared to a less flexible plant with minimal foliage (Coon 1997).

Petryk & Bosmajian (1975), Pasche & Rouve (1985), Poggi et al. (2009) determined the resistance of rigid vegetation. Li & Shen (1973), Petryk & Bosma jian (1975), Lindner (1982) are few of the researchers

Figure 1. Vegetation (Photo taken at Nif river, Turkey).

examined the resistance to flow of the vegetation using patterns of rigid cylinders basically treating plants as cylinders. Wu (2008) analyzed the flow resistance factors of non-submerged rigid vegetation. Pasquino et al. (2016) evaluated the vegetation resistance for fully submerged and rigid vegetation, for different hydraulic conditions and varying non-dimensional vegetation density. Kouwen & Uny (1973), Kouwen & FathiMoghodam (2000), Stephan & Gutknecht (2002) investigated the roughness of flexible vegetation. This type of vegetation could bend due to the flow velocity. The bending of plants decreases the height of the vegetation influencing the resistance and flow depth. For submerged flexible vegetation, three different configurations can be distinguished depending on the flow velocity and the plant characteristics (Kouwen et al. 1969, Gourlay 1970, Galema 2009). Vegetation that is erected and behave as rigid, that are subjected to a

waving motion and, the ones that keep their bended configuration all the time.

Kouwen (1992) developed a relationship between Manning's coefficient and product of flow velocity and hydraulic radius. Wu et al. (1999) worked with sub merged and non-submerged flexible vegetation under uniform flow conditions. For fully submerged vegeta tion they revealed that roughness coefficient tends to increase at low depths but then decrease to an asymp totic constant. Vargas-Luna et al. (2015a) listed the models to predict the resistance coefficient for emer gent conditions. In this list, he summarized the meth ods of Petryk & Bosmajian (1975), Raupach (1992), Ishikawa et al. (2003), James et al. (2004), Hoffmann (2004) and Kothyari et al. (2009). Jarvela (2004) pre sented a procedure for determining flow resistance by stiff and flexible woody vegetation in terms of friction factor f or Manning's n including both leafless and leafly bushes and trees. The procedure is limited to non-submerged flow and relatively low velocity con ditions which are often found in low-gradient stream valleys, flood plain and wetlands. Vargas-Luna et al. (2015b) conducted tests considering emergent and submerged vegetation using artificial grass, three dif ferent plant densities on non-erodible sand and gravel bed.They derive the properties of the array of cylinders that describe each vegetation configuration by apply ing Baptist's (2005) method. The equivalent cylinder density was found to vary according to bed roughness, even for the same flow regime and vegetation den sity. In spite of these studies, according to Augustijn et al. (2008), more data is required, to establish, which description performs best under flood of flow conditions. There is a need for a wide data set of flow experiments to evaluate the ranges of applicabil ity of vegetation resistance descriptions. Vargas-Luna et al. (2015a) provided an overview of existing mod els for the assessment of the effect of vegetation and concluded that additional research is needed to accu rately reproduce the effects, notwithstanding the great progress made in recent decades.

In evaluating the resistance, another important aspect is the description of the distribution, density and localization of vegetation. The number of plants or the vegetation density may vary significantly from place to place and hard to take into account in modelling. Most of the studies suppose that the vegetation is staggered and parallel or randomly distributed (Jarvela 2004, Wu 2008) whereas they grow especially along the river sides (Nehal et al. 2012). As a strong simplification of reality, it is commonly accepted to represent vegetation in open channels as a collection of identical stems and

static rigid cylinders (Jarvela 2004) however Vargas

Luna et al. (2014) claimed that the implications of this

assumption are still unknown.

This study constitutes a part of a more compre

hensive study covering the determination of flow

resistance as a response to variation of flow param

eters with time. Particularly, it is aimed in this study to

investigate the effect of vegetation distribution while

keeping the number of plants same. Figure 2. (a)&(b)View of the flume. 2 EXPERIMENTAL SET-UP The experiments are conducted in a rectangular recirculating flume of 0.18 m wide and 5 m long, in the Hydraulics Laboratory of Ege University, Department of Civil Engineering.The slope of the small scale flume was 0.007 (Fig. 2). First 75 cm of the flume is used to still the flow. Plastic artificial grasses were used in the experiments as vegetation cover. In the middle of the flume 1 m long section of the test area was covered with vegetation (Figs 3–5). The frontal area of the plants was obtained by scanning the pictures of plants taken and the areas of pixels were counted. The images of plastic strips are used to evaluate the projected area to the approaching flow. The average front area of each plant is 27 cm 2 with a standard deviation of 1 cm 2 . The projected area and diameter of the plants varies with its height. Variation of cumulative front area with depth is given in Figure 3. A significant number of flow resistance predictors treat vegetation simply as static rigid cylinders (Jarvela 2004). From this figure it is revealed that the variation of frontal area with vegetation height is different from a constant diameter cylinder; therefore it may be argumentative to assume the vegetation as a cylinder. The density of the vegetation which is defined as the number of stems per unit plain area m is 122 m –2 for

Figure 3. Relation between projected area and flow depth.

Figure 4. Distribution of vegetation (a) configuration 1, one

row, (b) and (c) configuration 2, two rows (side & plan views).

Figure 5. (a) Configuration 1, one row of vegetation, (b)

configuration 2, two rows of vegetation.

all experiments. Total number of flexible vegetation is

22 for all cases.

Three different configurations were studied namely

scenarios. In the first configuration all plants were

attached to bottom on one side of the flume (Fig. 4a

and Fig. 5a). In the second configuration, half of the

plants were taken out and re-attached to the other side

wall (Fig. 4b–c and Fig. 5b), so that the total number

of grasses was kept constant. In the last scenario, all

the plants were taken out and the flume left empty.

Water depths are obtained at seven locations before

and after the vegetation zone. Discharge is measured

via a scaled tank. Table 1. Characteristics of the scenarios. Q h_0 Fr # of runs Vegetation distribution l/s cm – – One row 0.38–2.93 0.80–10.05 0.04–1.36 31 Two rows 0.29–3.06 1.00–10.05 0.03–1.30 21 Empty 0.55–2.99 1.10–10.00 0.04–1.42 23 3 RESULTS Experiments performed are summarized in Table 1. For each configuration, different downstream flow depths and different discharges Q were applied and in total 75 runs were carried out with different Froude numbers Fr. Here h_0 is the entrance flow depth. The Reynolds number ($Re = 4VR/\nu$) is in the range of $3.6 \times 10^3 - 5.3 \times 10^4$. The water surface profiles along the flume for Q = 0.45 l/s are given in Figure 6, for illustrative purposes. The beginning and end of the vegetation zones are depicted by vertical green lines in Figure 6a and 6b, which corresponds to x = 125 cm and 225 cm of the flume. The figures show how water depths and velocity profiles are affected by the presence of vegetation. Water levels increase and vertical flow velocities decrease for the presence of vegetation, as expected. In all experiments, the steady flow conditions were observed.The uniform conditions were attained at only two experiments where the flume bed slope S_0 is equal to the slope of the energy grade line S_e . Approximately in 10% of 75 experiments, the slope of energy grade line is greater than 0.005. Therefore almost all experiments were conducted at non-uniform conditions. As such, Ishikawa et al. (2000), James et al. (2004), Liu et al. (2008) worked with uniform and Tanino & Nepf (2008), Ferreira et al. (2009), Kothyari et al. (2009) and Wang et al. (2015) worked with non-uniform flow. 4 HYDRAULIC ROUGHNESS The hydraulic resistance to flow through emergent vegetation was investigated in terms of coefficient of Chézy, Weisbach roughness coefficient and Manning's roughness coefficient. The formulas to predict these coefficients and to describe the channel roughness were first derived for pipes; however, they are now also used for describing resistance caused by vegetation (Galema 2009). French engineer Antoine Chézy (1769) established a conventional approach for describing the roughness of the bottom and side walls via the uniform flow formula: where V = velocity; C = coefficient of Chézy and R = hydraulic radius (Chow 1959).

Figure 6. Water surface profiles along the flume for

Q = 0.45 l/s (a) vegetation with one row, (b) two rows and

(c) empty flume.

A combination of the equation of Julius Weisbach

(derived in 1845) and the formula of Henry Darcy

(derived in 1858) resulted in the Darcy-Weisbach

equation, which can be written as

where g = gravitational acceleration; and f =Weisbach

roughness coefficient.

The Irish engineer Manning derived the formula for

open-channel uniform flow:

where n = Manning's roughness coefficient. There are

photos and reference tables for Manning's n values for

channels and closed conduits (Chow 1959).

Energy losses are obtained using the energy balance

and the continuity equations for all configurations. Figure 7. Variation of C coefficient of Chézy with Froude number. Figure 8. Variation of f Weisbach roughness coefficient with Froude number. The friction loss, H_f, is computed from the flume data using Bernoulli's equation (Jarvela 2002) where z is the elevation, α is the velocity distribution coefficient, and the subscripts 1 and 2 refer to upstream and downstream sections, respectively. The slope of energy grade line S_e is calculated by $S_e = H_f/L$ where L is the stream wise distance of the vegetation zone. The correction coefficient for the kinetic energy head is adopted as one (Wang et al. 2015). The variations of coefficient of Chézy, Weisbach roughness coefficient and Manning's roughness coefficient with Froude number are depicted for three scenarios on Figures 7–9, respectively. The maximum friction factor was obtained for low velocity and high flow depths as revealed by Jarvela (2002 & 2004). It is observed that for all methods of resistance descriptions, the scenario dealing with two rows gave the maximum resistance among the others. This is attributed to the restricted flow area by two rows of plants on both sides of the flume. On the other hand, the resistance of empty channel has the smallest one. Trend line of power type was imposed on the graphs of resistance coefficients and Froude number.

Figure 9. Variation of n Manning's roughness coefficient

with Froude number.

Figure 10. Comparison of coefficients determined in Figs.

7–9 for three configurations.

The coefficients of the power equations determined

in Figures 7–9 for three scenarios are compared in

Figure 10, verifying the hypothesis proposed above.

5 CONCLUSION

It is known that presence of vegetation could be consid

ered as a problem for channel flow capacity due to its

roughness. In order to understand the effects of plan

form areal configuration of flexible vegetation on flow,
a set of experiments were performed in the Hydraulics
Laboratory of the Department of Civil Engineering,
Ege University, in a rectangular flume which is 5 m
long, 18 cm wide and 20 cm deep with transparent
sides. Experiments were conducted with three scenar
ios and two different spatial distributions of vegetation
keeping the total number of grass same. In the first
scenario grasses positioned on one half of the flume
and in the second one double sided. In the third sce
nario the flume is emptied. For each scenario different
downstream flow depths and different discharges were
applied. In total 75 runs were performed with different
Froude numbers. Energy losses were calculated using
the energy balance and the continuity equations for all
scenarios.The variations of coefficient of Chézy,Weis
bach roughness coefficient and Manning's roughness
coefficient with Froude number are obtained and a
power equation was imposed to the data. It is observed

Kouwen, N. 1992. Modern approach to design of grassed
channels. Journal of Irrigation and Drainage Engineer
ing, 118(5): 733–743.
Kouwen, N. & Fathi-Moghadam, M. 2000. Friction factors
for coniferous trees along rivers. Journal of Hydraulic
Engineering, 126(10): 732–740.

Kouwen, N., Unny, T.E. & Hill, H.M. 1969. Flow retar
dance in vegetated channels, Journal of 421 Irrigation
and Drainage Division-ASCE 95(IR2), 329–342.
Kouwen, N. & Unny, T. 1973. Flexible roughness in open
channels. Journal of Hydraulic Div., 99(5), 713–728.
Li, R.M. & Shen, H.W. 1973. Effect of tall vegetations on
flow and sediment. Journal of Hydraulics Division,ASCE,
99(5). 793–814.
Lindner, K. 1982. Der Strömungswiderstand von PflanzenBe
standen. Mitteilungen Leichtweiss-Institut für Wasserbau,
No:75. TU Braunschweig.
Liu, D., Diplas, P., Fairbanks, J.D. & Hodges, C.C. 2008.
An experimental study of flow through rigid vegetation,
Journal of Geophysical Research-Earth Surface, 113(F4),
F04015.
Nehal L., Yan, Z.M., Xia, J.H. & Khaldi, A. 2012. Flow
through non-submerged vegetation: a flume experiment
with artificial vegetation, Sixteenth International Water
Technology Conference, IWTC 16 2012, Istanbul, Turkey.
Pasche, E. & Rouvé, G. 1985. Overbank flow with veg
etatively roughened flood plains. Journal of Hydraulic
Engineering. 111(9): 1262–1278.
Pasquino, V., Gualtieri, P, & Doria, G.P. 2016. On Evalu
ating Flow Resistance of Rigid Vegetation Using Clas
sic Hydraulic Roughness at High Submergence Levels:

An Experimental Work. Hydrodynamic and Mass Trans

port at Freshwater Aquatic Interfaces, Part of the series

GeoPlanet: Earth and Planetary Sciences, 269–277.

Petryk, S., & Bosmajian, G., 1975. Analysis of flow through

vegetation, Journal of the Hydraulics Division, 101,

871–884. Poggi, D., Krug, C. & Katul, G.G. 2009. Hydraulic resistance of submerged rigid vegetation derived from first-order closure models. Water Resources Research, 45, W10442, doi:10.1029/2008WR007373. Raupach, M.R., 1992. Drag and drag partition on rough surfaces. Boundary-Layer Meteorology. 60, 374–396. Stephan, U. & Gutknecht, D. 2002. Hydraulic resistance of submerged flexible vegetation. Journal of Hydrology, 269(1–2), 27–43. Tanino, Y. & Nepf, H.M. 2008. Lateral dispersion in random cylinder arrays at high Reynolds number, Journal of Fluid Mech. 600, 339–371. Vargas-Luna A., Crosato, A. & Uijttewaal, W.S.J. 2015a. Effects of vegetation on flow and sediment transport: comparative analyses and validation of predicting models. Earth Surface Processes and Landforms. 40(2), 157–176. Vargas-LunaA., Crosato,A. Calvania, G. & Uijttewaal,W.S.J. 2014. Mimicking the effects of vegetation in laboratory setups. 10th ISE, Trondheim, Norway. Vargas-Luna A., Crosato, A. Calvania, G. & Uijttewaal, W.S.J. 2015b. Representing plants as rigid cylinders in experiments and models. Advances in Water Resources. doi:10.1016/j.advwatres.2015.10.004 Wang,W.J., Huai, W.X., Thompson, S. & Katul, G.G. 2015. Steady nonuniform shallow flow within emergent vegetation. Water Resources Research, 51, 10047–10064, doi:10.1002/2015WR017658. Wilson, C.A.M.E. 2007. Flow resistance models for flexible submerged vegetation. Journal of Hydrology, 342(3-4): 213–222. Wu, F.C., Shen, H.W., & Chou, Y.J. 1999. Variation of roughness coefficients for unsubmerged and submerged vegetation. Journal of Hydraulic Engineering, 125(9), 934–942. Wu, F.S. 2008. Characteristics of flow resistance in open channels with non-submerged rigid vegetation. Journal of Hydrodynamics, Ser. B. 20(2), 239–245. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Monthly reservoir operating rules generated by implicit stochastic optimization and self-organizing maps

C.A.S. Farias, E.C.M. Machado & L.N. Brasiliano

Federal University of Campina Grande, Pombal – PB, Brazil

ABSTRACT: This paper reports on the application of Implicit Stochastic Optimization (ISO) and Self

Organizing Maps (SOM) to determine monthly operating policies for a reservoir system located in semiarid

Brazil. Three steps define the ISO-SOM model: (1) generation of monthly inflow scenarios; (2) determinist

optimization of the system operation under the generated inflow scenarios; and (3) development of monthly

rules by means of SOM, which consists of conditioning allocation coefficients on initial reservoir storage and

current inflow. The study demonstrates the superiority of ISO-SOM model over the standard rules of reservoir

operation, especially during drought spells.

1 INTRODUCTION

According to Tundisi (2008), some reasons for the so called "water crisis" of the twenty-first century are: increasing demands for water; changes in water avail ability; intensification of climate extremes; and lack of articulated initiatives and government actions towards water governability and environmental sustainability.

In semiarid Brazil, the sustainable management of water resources is a complicated task because of the natural hydrological conditions (e.g. high evapora tion rates, scarce and uneven space-time distribution of rainfalls and limited groundwater availability).

Such environment suffers periodically from water scarcity, what compromises not only social and eco nomic development but also water and food security (Carneiro & Farias, 2013; Farias et al., 2015).

The use of predetermined operating rules seems to be a common and practical solution to manage water systems subject to drought spells. In a water supply reservoir, for example, such rules may be derived by applying optimization procedures and linear regres sion equations (Loucks & Beek, 2005).

The Implicit Stochastic Optimization (ISO) proce dure consists of optimizing the system operation under several inflow scenarios and using the optimal out comes to define water allocation policies. Usually, linear regression models relate initial reservoir stor ages and predicted inflows to optimal releases (Farias et al., 2011; Loucks & Beek, 2005). Unlike the use of regression equations, we propose the application of a Self-Organizing Map (SOM) model as an attempt to extract possible nonlinear trends among the variables of the process.

As stated by Kohonen (1982), a SOM is a bidi

mensional matrix of features capable of representing multidimensional sets of data. The basic principle consists of grouping data vectors according to their similarities in a map, which may be later of use to pattern classifications and analyses. SOM are a type of unsupervised Artificial Neural Networks based on a competitive training approach

(Haykin, 1999). Although reservoir operation studies on SOM are rare, other water engineering fields have successfully benefited from this emerging tool. García & González (2004), for instance, implemented a SOMbased model to estimate and monitor the different states of a wastewater treatment plant. Using data from weather stations in UK and India, Adeloye et al. (2011) verified the superiority of a SOM model over other empirical methods to estimate reference crop evapotranspiration. More recently, Farias et al. (2015) compared several SOM structures in order to set a runoff-erosion model for a semiarid land of Brazil. 2 STUDY AREA AND DATA The water system adopted for this study is composed of the Coremas and Mãe D'Água reservoirs, which are located in Piancó River basin, Brazil. An open channel with a flow capacity of 12 m 3 /s connects Coremas and Mãe D'Água reservoirs. In conformity with Celeste et al. (2009), we simplified the mathematical implementation of Coremas-Mãe D'água system by modelling it as an equivalent reservoir. As a result, the sum of active storages of both reservoirs becomes the active storage of the equivalent reservoir. Similarly, the flows into the equivalent reservoir matches the sum of flows into each individual reservoir. Table 1 shows the characteristics of equivalent and individual reservoirs.

Figure 1. Location of Piancó River basin.

Table 1. Characteristics of equivalent and individual

reservoirs. Dead Maximum Active storage storage storage

Reservoir (hm 3) (hm 3) (hm 3)

Coremas 32.0 720.0 688.0

Mãe D'Água 14.8 638.7 623.9

Equivalent Reservoir 46.8 1358.7 1311.9

Source: Celeste et al. (2009)

Table 2. Water demands in Coremas-Mãe D'Água water

system. Domestic Fish consumption Irrigation farming Total

Month (m 3 /s) (m 3 /s) (m 3 /s) (m 3 /s)

January 2.28 8.12 0.049 10.45

February 2.28 7.45 0.049 9.78

March	2.28	6.77	0.049	9.10
April	2.28	6.68	0.049	9.01
May	2.28	6.51	0.049	8.84
June	2.28	6.64	0.049	8.97
July	2.28	7.11	0.049	9.44
August	2.28	7.92	0.049	10.25
September	2.28	8.40	0.049	10.73
October	2.28	8.70	0.049	11.03
November	2.28	8.72	0.049	11.04
December	2.28	8.21	0.049	10.54

Source: .Lima (2004)

Streamflow data are available for three gauging sta

tions in Piancó River basin (Fig. 1). Monthly means

for evaporation and precipitation are at disposal in the

work of Lima (2004). The main water demands in the system are domestic consumption, industrial and irrigation use, and fish farming. Coremas reservoir also explores a small hydropower plant. Table 2 lists the monthly demands assumed for this study. 3 ISO-SOM MODEL The ISO procedure comprises three basic steps: (1) Generate M synthetic monthly scenarios of reservoir inflows; (2) Optimize the system operation for all M scenarios using a deterministic optimization model; and (3) Use the ensemble of optimal outcomes to develop monthly operating rules. In this study, coefficients of allocation $\alpha(t)$ for each month index t are related to initial reservoir storages $S(t-1)$ and reservoir inflows $I(t)$ during the current time period by means of SOM Consequently, with information on initial reservoir storage and inflow for the current month, it is possible to determine the water release for a specific month. The reservoir release $R(t)$ and water demand $D(t)$ defines the coefficient of allocation as shown in Equation 1. 3.1 Stochastic generation of inflows The success of an ISO procedure depends on the existence of long series of hydrological records, which is not always possible. A practical solution to overcome such difficult is to carry out stochastic simulations of hydrological variables. For this, the simulation model

Figure 2. SOM structure with 5 × 5 neurons and a neighborhood radius of three steps. Source: Adapted from Farias & Santos

(2014)

must produce data with statistical properties similar to

the observed values.

The model used for the monthly generation of

reservoir inflows is the Fragment Method (FM).

Such method has been effectively applied to semiarid

regions and more detailed information on its function

ing may be accessed in the works of Carneiro & Farias

(2013), Celeste et al. (2007), and Svanidze (1980).

3.2 Optimization of reservoir operation

The determinist model assumes that the main goal of

the system operation is to find water releases that best

satisfy the total demands without compromising the

system, as shown in Equation 2.

where Z = objective function; and N = operating hori

zon.

The water balance (Eq. 3) relates water release, stor

age, inflow, precipitation, evaporation and spill at each

period.

in which S(t) = reservoir storage at the end of month

t; S(0) = initial reservoir storage; I (t) = reservoir inflow; $P(t)$ = precipitation volume over lake sur

face; $E(t)$ = evaporation volume over lake surface; and

$Sp(t)$ = spill that eventually might occur. The physical limitations of the system define lower and upper bounds for releases, storages and spill, as shown in Equations 4–6. where S_{max} and S_{min} are the maximum and minimum reservoir storages, respectively. 3.3 Operating rules The ISO-SOM model is used to produce twelve operating rules, one for each month of the year. A SOM contains single units known as neurons, which auto-organize in a manner that preserves neighboring relationships. According to Beale et al. (2012), the neurons in a SOM are sets of parameters (weights) which are adjusted in order to recognize input vectors. The SOM network is composed of two layers: a multidimensional input layer and a bidimensional output layer. In the later, neurons compete in order to define a winner. Each component of an input vector connects the weights of all neurons in the bidimensional layer. In this study, the input vectors are formed by three elements: initial storage $S(t-1)$, inflow $I(t)$, and coefficient of allocation $\alpha(t)$. The output layer contains hexagonal neurons with three weights, one for every component in input vectors. Figure 2 shows a SOM structure with 5×5 neurons and a neighborhood radius of three steps.

As shown in Equation 7, Euclidian distances Ed_i

between input vectors and output neurons i control the

calibration of SOM networks.

in which j = vector component index; p = total num

ber of weights in each output neuron and elements in

input vector; b = number of neurons in the bidimen

sional layer; $x(j)$ = j-th component of the input vector

x; and $w_i(j)$ = j-th weight component of the i-th output

neuron.

The winning neuron is the neuron i for which the

Euclidean distance to input vector x is the lowest.

The weights associated with this neuron i* and neigh bor neurons in a certain neighborhood radius Vi* are then updated by the Kohonen rule, which is available in Beale et al. (2012). The application of this rule modifies the weights of the winning neuron and its neighbors in order to decrease Euclidean distances. As a result, the map of neurons may be used later to classify similar vectors.

In this study, the trainings occur in batch mode. In such mode, the search for winning neurons is carried out for each input vector (sample) and then the weight vector is moved to the average position of all input vectors for which it is a winner or neighbor of a winner. After several presentations of the data set, the weights tend to stabilize (Beale et al., 2012).

In this approach, a number of 100 presentations of the whole data set limits the network trainings. The radius of the neighborhood starts with three steps and uniformly decreases to the unit value. This choice is a way to organize the neuron weights in the input space consistently with neuron positions in the dimensional grids (Beale et al., 2012).

After calibrations, the SOM networks can estimate allocation coefficients for new input data. For this, we consider the allocation coefficient component as missing in the input vector and follow three steps:

(1) calculate the Euclidean distances between the input

vector and weights of output neurons disregarding

the allocation component;

(2) determine the winning neuron based on the lowest

Euclidean distance; and

(3) assume the weight component of the winner neuron

connected to the missing value of the input vector

as the estimation.

The models developed in this study were imple

mented in MathWorks' MATLAB R2012a (Beale et

al., 2012).

4 RESULTS AND DISCUSSION

4.1 Stochastic generation of inflows

The data used for calibrating the FM comprises the

sum of monthly streamflows from Piancó, Emas and Table 3. Correlations between observed and calculated coefficients of allocation for the training dataset. Month Correlation coefficient January 0.96 February 0.95 March 0.93 April 0.94 May 0.96 June 0.93 July 0.95 August 0.96 September 0.94 October 0.94 November 0.94 December 0.94 Aguiar gauging stations during the period from 1964 to 1988. The model and inflow series used in this research are the same ones implemented and successfully tested by Carneiro & Farias (2013). 4.2 ISO-SOM operating rules The initial storage of the equivalent reservoir was set to the maximum storage. The ISO-SOM model was run under M = 20 sequences of N = 1248 months of FM-based inflows. The first and last two years of each sequence M were suppressed to avoid problems with boundary conditions. This provided a total of 24,000 months (20 sequences of 1,200 months) of optimal reservoir releases. The reservoir initial storages, current inflows and allocation coefficients from January to December were grouped and submitted to the SOM model for obtaining 12 operating rules, one for each month. Each SOM was calibrated with b = 25 neurons (5 × 5) in the bidimensional layer as shown in Fig. 2. Table 3 shows the correlations between observed and calculated coefficients of allocation for each month considering the training dataset. The correlations were equal to or greater than 0.93 for all 12 rules, indicating that those data are highly correlated. Figure 3 depicts the SOM component planes for all months. The color scales represent the values of neurons according to their position in the map. Yellow zones correspond to higher values of neuron weights while black zones indicate lower values. Analysis of Figure 3 confirms that high values of initial storage and current inflow matches high coefficients of allocation. Moreover, the maps of components demonstrate that both the season and reservoir initial storage play important roles in deciding the coefficient for water allocation. The performance of the ISO-SOM model was investigated considering the operation of the reservoir for 10 new FM-based simulations of inflows, each one with 24 years. As in the ISO-SOM model calibration process, the two first and last years were eliminated,

Figure 3. Component maps for all months.

resulting in a total of 20 years (240 months) of

operation for each new simulation.

The ISO-SOM model results were compared to

those obtained by the application of the deterministic

model under perfect forecasts of reservoir inflows.The

operation using the deterministic model gives us the

“optimal releases” to be employed since it has knowl

edge of future monthly inflows for the whole operating

horizon.

In addition, the Standard Linear Operating Pol

icy, also known as SLOP, was applied for comparison

purposes.The SLOP, which is one of the simplest oper

ating rules, states that when the available water is equal

to or less than the demand, all stored water is released.

When the available water exceeds the demand, the demand is met and the surplus is accumulated in the reservoir until its maximum storage is reached and spillage starts to occur (Loucks et al., 1981).

A vulnerability index U , assumed as the function shown in Equation 8, was used for all simulations and

Table 4. Vulnerability index for 10 simulations of 240 months.

Simulation	Deterministic	SLOP	ISO-SOM model
SIM#01	0.02	0.07	0.04
SIM#02	0.05	0.10	0.08
SIM#03	0.02	0.06	0.04
SIM#04	0.10	0.19	0.16
SIM#05	0.08	0.15	0.13
SIM#06	0.01	0.01	0.01
SIM#07	0.06	0.12	0.09
SIM#08	0.02	0.05	0.03
SIM#09	0.02	0.06	0.04
SIM#10	0.00	0.00	0.00

Figure 4. Reservoir storage results for the reservoir operation in SIM#01. models of this study and the results are presented in Table 4. The vulnerability index denotes the magnitude of system failures to attend water demands for the whole operating horizon N . The vulnerability varies from zero to one, with “zero” indicating that the demand was fully attended and “one” demonstrating that there was no water allocation at all. As a result, the higher is the vulnerability indicator; the more vulnerable is the rule. Analysis of Table 4 shows that the ISO-SOM model was less vulnerable than SLOP and more vulnerable

than the deterministic model for eight of the 10 simulations. Figure 4 illustrates the reservoir storage behavior of all models for SIM#01. Investigation of the results shown in Figure 4 evidences that the deterministic model tries to reserve the stored water to use in future shortage periods. In periods of water shortage, it is possible to suggest that the ISO-SOM policies try to imitate the reservoir storage behavior observed in the operation with the deterministic model, which is an optimization under perfect forecast. 5 CONCLUSION We coupled ISO and SOM to derive monthly operating rules to a water system in semiarid Brazil. The calibration of the reservoir operating rules were satisfactory for the proposed SOM structure of 5 × 5 neurons. Moreover, the generated maps of components

enabled a detailed analysis and understanding of the

specific reservoir operation.

The outcomes indicate that the ISO-SOM model

is less vulnerable to water shortages than the stan

dard rules of reservoir operation. As conclusion, we

believe that the combination of ISO with emerging

tools such as SOM may be a promising strategy for

reservoir operation.

ACKNOWLEDGMENT

The authors gratefully acknowledge the Brazilian

National Council for Scientific and Technological

Development (CNPq) for financial support (Process

n. o 477838/2013-8).

Adeloye, A. J., Rustum, R., & Kariyama, I. D. 2011. Koho

nen self-organizing map estimator for the reference crop

evapotranspiration. Water Resources Research, 47(8).

Beale, M., Hagan, M., & Demuth, H. (2012). Neural Network

Toolbox 7.0.3: User’s Guide. Natick: The MathWorks Inc.

Carneiro, T. C., & Farias, C. A. S. 2013. Otimização estocás

tica implícita e redes neurais artificiais para auxílio na

operação mensal dos reservatórios Coremas – Mãe d’

Água. Revista Brasileira de Recursos Hídricos, 18(4),

115–124.

Celeste, A. B., Curi, W. F., & Curi, R. C. 2007. Análise de

métodos para geração sintética de vazões. In XVII Sim

pósio Brasileiro de Recursos Hídricos (pp. 1–18). São

Paulo – SP.

Celeste, A. B., Curi, W. F., & Curi, R. C. 2009. Implicit

stochastic optimization for deriving reservoir operating

rules in semiarid Brazil. Pesquisa Operacional, 29(1),

223–234. Farias, C. A. S., Bezerra, U. A., & Silva Filho, J. A. 2015. Runoff-erosion modeling at micro-watershed scale: a comparison of self-organizing maps structures. Geoenvironmental Disasters, 2(1), 2–14. Farias, C. A. S. de, & Santos, C. A. G. 2014. The use of Kohonen neural networks for runoff–erosion modeling. Journal of Soils and Sediments, 14(7), 1242–1250. Farias, C. A. S. de, Santos, C. A. G., & Celeste, A. B. 2011. Daily reservoir operating rules by implicit stochastic optimization and artificial neural networks in a semi-arid land of Brazil. IAHS Publications, 347, 191–197. Garcìa, H. L., & González, I. M. 2004. Self-organizing map and clustering for wastewater treatment monitoring. Engineering Applications of Artificial Intelligence, 17(3), 215–225. Haykin, S. 1999. Neural networks: a comprehensive foundation (2nd Ed.). Upper Saddle River: Prentice Hall. Kohonen, T. 1982. Self-organized formation of topologically correct feature maps. Biological Cybernetics, 43(1), 59– 69. Lima, C. A. G. 2004. Análise e sugestões para diretrizes de uso das disponibilidades hídricas superficiais da bacia hidrográfica do Rio Piancó, situada no Estado da Paraíba. Universidade Federal de Campina Grande. Loucks, D. P., & Beek, E. Van. 2005. Water resources systems planning and management: an introduction to methods, models and applications. Turin: United Nations Educational, Scientific and Cultural Organization. Loucks, D. P., Stedinger, J. R., & Haith, D. A. 1981. Water resources systems planning and analysis. Englewood Cliffs: Prentice Hall. Svanidze, G. G. 1980. Mathematical modeling of hydrologic series (for hydroeletric and water resources computations). Water Resources Publications. Fort Collins. Tundisi, J. G. 2008. Recursos hídricos no futuro: problemas e soluções. Estudos Avançados, 22(63), 7–16. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Comparison of PIV measurements and CFD simulations of the velocity field

over bottom racks

Luis G. Castillo, Juan T. García, José M. Carrillo & Antonio Vigueras-Rodríguez

Civil Engineering Department, Hidr@m I+D+i Group, Universidad Politécnica de Cartagena, Spain

ABSTRACT: In this work, the comparison of the velocity field over a bottom rack system measured by

Particle Image Velocimetry (PIV) and simulated with numerical simulations (ANSYS CFX v14.0) is presented.

Laboratory measurements are taken in a physical device located in the Laboratory of Hydraulic Engineering of

the Universidad Politécnica de Cartagena (Spain). Velocity and pressure coefficients of the energy equation are

obtained and used to evaluate the water profile along the racks. Pressure distribution along the flow depth is

presented for several distances along the rack. Pressure results are compared with the pressure deviation terms

from hydrostatic pressure profile proposed by several authors.

1 INTRODUCTION

Bottom intake systems, made by racks disposed lon

gitudinally to flow and located at streambed, are used to derive flood flows from ephemeral gullies in semi arid regions. The shape and spacing between the bars that constitute the rack have influence in the derived flow per unit length. Leading, therefore, to different discharge coefficient values (Orth et al., 1954). The intake system is a spatially-varied flow with decreas ing discharge, in which the curvature of the water profile and the streamlines creates a non-hydrostatic pressure distribution over the bottom rack. Following previous studies of Mostkow (1957), Righetti and Lan zoni (2008) verified the relation between the angle of streamlines with the plane of the rack and the discharge coefficient. The streamlines slope also influences in the direction of the drag force that water exerts on solids, defining areas of preferential deposition of solids over the racks (Castillo et al., 2013a, 2013b, 2014, 2015).

Several researchers proposed analytical solutions of the continuity and momentum equations in the vertical plane over the rack. Nakagawa (1969) used a a lin eal profile for the horizontal velocity component with regard to its average in each section, while Castro Orgaz and Hager (2011) estimated it as a constant. These solutions provide pressure distribution and its deviation from hydrostatic values.

Mostkow (1957) considered two-dimensional equa

tions of momentum and continuity.

Common solutions to estimate the water profile

and the derived flow along bottom intakes consider

frictionless irrotational flow with hydrostatic pressure

distributions (Garot, 1939; De Marchi, 1947; Noseda,

1956). For the horizontal rack case where h is the flow depth, H the energy head considered constant, m the void ratio, and C qh the discharge coefficient as a function of the flow depth. Curvilinear flow over bottom racks and slots have been experimentally characterized in laboratory by several authors using pressure measurements, and obtaining velocity and pressure coefficients (Mostkow, 1957; Nakagawa, 1969; Nasser et al., 1980). 2 OBJECTIVES The definition of velocity and pressure fields along bottom systems and its influence in derived flow are of interest. In this work, a Particle Image Velocimetry (PIV) system is used to define the 2D velocity field in a vertical plane located in the space between bars. Results are compared with computational fluid dynamic (CFD) simulations (ANSYS CFX v14.0). Velocity and pressure coefficients, α and λ respectively, are defined and used to evaluate the water profile and the derived flow per unit length. For that purpose, the following equation, obtained from frictionless energy equation, has been used: 3 MATERIAL AND METHODS 3.1 Physical device An intake system located at the Hydraulic Laboratory of the Universidad Politécnica de Cartagena (Spain) has been used. It consists of a 5.00 m long and 0.50 m wide approximation channel, a rack with different

Figure 1. Scheme of bars position.

Table 1. Geometric characteristic of racks.

Experiment A B C

Spacing between 5.70 8.50 11.70

bars, b 1 (mm)

Void ratio 0.16 0.22 0.28

m = b 1 b 1 +30

slopes (from horizontal to 33%), a discharge chan nel, and the channel to collect derived water. Three different racks, with 0.90 m length, are available. All of them are made of aluminium bars with T profiles (T 30/25/2 mm). Bars are disposed longitudinally to the inlet flow. The differences be-tween the racks are the spacing between bars, so different void ratios are available (Figure 1).

Table 1 summarizes the geometric characteristics of each rack.

In this work, the rack with void ratio m = 0.28 in the horizontal position was used. Figure 2 shows the intake system in the Hydraulic Laboratory.

3.2 PIV equipment

Velocity field was measured with a PIV system com posed by a high-speed camera Motion Pro HS-3, 75 mm focal length objective, lens aperture f/11, 520 × 520 pixel resolution, 8 bits → 255 shades and a distance from the camera to stream recorded of 0.50 m. Recording window dimensions are 9x9 cm.

The laser is an Oxford Laser whose configura tion is: pulse = 10 μs; beam width = 5.5 mm; power peak = 200 W; delay = 30 μs; wavelength = 808 nm.

The temporal increment between frames is t = 1/600 s; so the ratio = 0.00017 meter/pixel.

Duration of each test was about 12.5 seconds.

Flow was seeded with polyamide particles of 50 μm

size. Frames were analysed in consecutive pairs by

cross-correlation in an interrogation area of 64 × 64

pixel with sub-windows of 32 × 32 pixel (Thielicke &

Stamhuis, 2014).

3.3 Numerical simulations

A Computational Fluid Dynamics simulation of the

intake system with ANSYS CFX v14.0 was also Figure 2. Intake system physical device. used. Previous works demonstrated the suitability of this code to solve the flow through an intake system (Castillo and Carrillo, 2012; Castillo et al., 2014, 2015). CFD codes solve the differential ReynoldsAveraged Navier-Stokes (RANS) equations of the phenomenon in the fluid domain, retaining the reference quantity in the three directions for each control volume identified. The equations for conservation of mass and momentum may be written as: where i and j are indices, x_i represents the coordinate directions (i, j = 1 to 3 for x, y, z directions, respectively), ρ the flow density, t the time, U the velocity vector, p the pressure, u'_i presents the turbulent velocity in each direction (i = 1 to 3 for x, y, z directions, respectively), μ the molecular viscosity, S_{ij} the mean strain-rate tensor, and $-\rho \overline{u'_i u'_j}$ the Reynolds stress. Eddy-viscosity turbulence models consider that such turbulence consists of small eddies which are continuously forming and dissipating, and in which the Reynolds stresses are assumed to be proportional to mean velocity gradients. The Reynolds stress may be related to the mean velocity gradients and eddy viscosity by the gradient diffusion hypothesis: with k being the eddy viscosity or turbulent viscosity, μ_t the eddy viscosity or turbulent viscosity and δ the Kronecker delta function. The k-ω based Shear-Stress-Transport (SST) turbulence model was selected to complement the numerical solution of the Reynolds-averaged Navier-Stokes

equations (RANS). To solve the two-phase air-water,

the homogeneous model was used. The fluid domain is

divided into control volumes, which must satisfy the balance of the governing equations. The total num ber of elements used in the simulations was around 350,000 elements, with 0.004 m length scale near the rack.

For simplicity, it was considered that all the longitu dinal bars work in the same mode in the intake system. For this reason, the domain fluid considers three bars and two spacing between bars. Symmetry conditions were used in the central plane of the extreme bars. The model boundary conditions correspond to the flow at the inlet condition (located 0.50 m upstream of the rack), the upstream and downstream water levels and their hydrostatic pressures distributions. In the bottom of the water collected channel, opening boundary condition were used. It has been assumed that the free surface is on the 0.5 air volume frac tion. To judge the convergence of iterations in the numerical solution, we monitored the residuals. The solution is said to have converged in the iterations if the scaled residuals are smaller than fixed values rang ing between 10^{-3} and 10^{-6}. In this work, the residual values were set to 10^{-4} for all the variables (Castillo et al. 2016).

4 RESULTS

4.1 Velocity field

Particle Image Velocimetry (PIV) allowed us to cal

culate velocity field and streamlines along the flow

over bottom racks. The orientation of the laser light

sheet is vertical and streamwise at the centreline of

the screen. Values are compared with numerical sim

ulations. Figures 3–5 show the velocity vector field,

together with the streamlines and the free surface

flow profile are presented for approximation flow of

q_1 = 77.0; 114.6 and 138.8 l/s/m, void ratio m = 0.28

and horizontal rack slope. Data consists in a steady

state test. Duration of each test was about 12.5 seconds.

Free surface is measured in lab, with a good agree

ment with CFD numerical simulation (Castillo et al.,

2014, 2016). Velocities and streamlines show a good

agreement between measured and simulated values.

4.2 Velocity and pressure coefficients

The coefficients of velocity (α) and pressure (λ) of

the energy equation can be obtained, by numerical

integration, from the following equations:

where U_i is the horizontal component of the vector

velocity, U the velocity module of the cross section,
Figure 3. Velocity field and streamlines measured with PIV and simulated with CFD for rack with m = 0.28, horizontal slope and approximation flow, q_1 = 77.0 l/s/m. Figure 4. Velocity field and streamlines measured with PIV and simulated with CFD for rack with m = 0.28, horizontal slope and approximation flow, q_1 = 114.6 l/sm. Figure 5. Velocity field and streamlines measured with PIV and simulated with CFD for rack with m = 0.28, horizontal slope

and approximation flow, q_1 = 138.8 l/sm. A the area of flow, q the specific flow across the considered section, y the vertical coordinate of the point in the cross section, and p the pressure in the point in which the y value is considered. In Figures 6 and 7, the coefficients of velocity and pressure from equations 6 and 7 are shown for different cross sections located in different distances to the beginning of the rack (0.00, 0.05, 0.10, 0.20, and 0.30 m), as well as for three specific approximation flows (77.0, 114.6, and 138.8 l/s/m). Coefficients presented in Figure 6 and 7 are obtained as a result of the proportional weight of the areas located over and between the longitudinal bars of the rack. From these coefficients, the Equation 2 can

Figure 6. Velocity coefficient of the energy equation, α, in cross sections located X distances from the beginning of the rack, and for three specific approximation flows.

Figure 7. Pressure coefficient of the energy equation, λ, in cross sections located X distances from the beginning of the rack and for three specific approximation flows.

be numerically solved using the fourth-order Runge–Kutta algorithm. To solve the system, the equation of flow derived is required:

The system of Equations 2 and 8 is equivalent to the solution of two ordinary differential equations with the unknown quantities h(x) and q(x).

At the inlet section, two boundary conditions are considered: the inlet specific flow q and the ini tial water depth h (being energy estimated as critical section).

Along the rack, the values of α, dα/dx, λ, and dλ/dx can be adjusted to exponential functions, expressed as functions of the x coordinate.

The discharge coefficient value is obtained from

(Noseda, 1956): Figure 8. (a) Comparison of the flow profile over the bottom rack along the rack solved with equations 2 and 8, and with laboratory measurements. Figure 8. (b) Comparison of the flow derived along the rack solved with equations 2 and 8, and with laboratory measurements. where l is the interaxis distance. In this case, the interaxis distance is 0.0417 m. The numerical results for h(x) and for the derived flow q d obtained are in agreement with the laboratory measurements. Figures 8a and 8b shows the results obtained for the specific flow of 114.6 l/s/m. 4.3 Pressure head along the rack The curvature of streamlines in the flow leads to pressure deviations from hydrostatic conditions. CastroOrgaz and Hager (2011) proposed an expression to calculate this deviation: where p is the pressure deviation, g the gravitational acceleration, q ′ = dq/dx the derived flow (Equation 8), h ′ = dh/dx the slope of the surface of flow, and h ′′ = d 2 h/dx 2 the curvature of flow profile.

Figure 9. Pressure head calculated from velocity field and

equation 10 compared with pressure head computed with

CFD in a cross section located 0.00 m from the beginning

of the rack.

Figure 10. Pressure head calculated from velocity field and

equation 10 compared with pressure head computed with

CFD in a cross section located 0.05 m from the beginning

of the rack.

From the field of velocities along the flow, in

Figures 9–12 the three terms on the right side of

Equation 10 are calculated and compared with the

pressure head computed with CFD, p CFD , in several

cross sections and along the flow depth for the case of q = 114.6 l/sm. Equation 11 shows the terms on the right side of Equation 10 defined as �p I , �p II , �p III and �p:

Some differences are observed between the terms of pressure head computed with CFD, p CFD , and the term (h-y +�p) computed from velocity field and equation 10 (Castro-Orgaz and Hager, 2011). These differences are significant in the lower zones of the Figure 11. Pressure head calculated from velocity field and equation 10 compared with pressure head computed with CFD in a cross section located 0.10 m from the beginning of the rack. Figure 12. Pressure head calculated from velocity field and equation 10 compared with pressure head computed with CFD in a cross section located 0.20 m from the beginning of the rack. flow, near to the bottom rack, while 2–3 cm above the bottom rack, values are very similar. From the Euler equation in the vertical direction and the continuity equation, the term�p (Equation 10) was obtained by integration in the vertical direction. The zones near the bottom rack are characterized with significant shear stress due to the increment of turbulence generated by a relevant transversal derivative of the vertical velocity within that area. Actually, vertical velocity has to change from significant values in the centre of the spacing between bars, to near null values close to the rack. Thus as a first approximation, the term of viscous stresses that would appear in the vertical Euler equation, $\upsilon_t \left(\frac{\partial^2 U_y}{\partial x^2} + \frac{\partial^2 U_y}{\partial y^2} + \frac{\partial^2 U_y}{\partial z^2} \right)$, has been calculated, resulting in a new equation system:

where U_x and U_y are the horizontal and vertical velocity components, respectively, and υ_t the kinematic eddy viscosity.

Neglecting terms of $\frac{\partial^2 U_y}{\partial x^2} + \frac{\partial^2 U_y}{\partial y^2}$, a numerical integration in the vertical direction has been done. Kinematic eddy viscosity divided by gravity acceleration is in the order of 10^{-4} ms, while the term $\partial^2 U_y / \partial z^2$ shows values in order of 10^3 $m^{-1} s^{-1}$. Integrating in the vertical direction, values of 10^{-2} m are obtained in the bottom part of the flow.These results are in agreement with the differences between p_{CFD} and the term $(h - y + \Delta p)$ showed in Figures 9–12.

5 CONCLUSIONS

The definition of the velocity field through the bottom racks is of importance to evaluate the derivation capacity and clogging phenomena over intake systems. In this work, Particle Image Velocimetry (PIV) laboratory measurements and Computational Fluid dynamics simulations (CFD) have been used to obtain the velocity field.

The knowledge of the velocity and the pressure coefficients in the energy equation, allows to define the flow profile and the derivation flow with a good agreement to the values measured in laboratory.

In a first approximation, pressure heads computed with CFD, show differences with empirical methods proposed that does not take into account turbulent viscous stresses.

Further experimental measurements and CFD simulation are required to improve the knowledge in curvilinear flows with decreasing discharge in bottom intake systems.

ACKNOWLEDGMENTS

The authors are grateful for the financial support received from the Seneca Foundation of Región de Murcia (Spain) through the project “Optimización de los sistemas de captación de fondo para zonas semiári das y caudales con alto contenido de sedimentos. Definición de los parámetros de diseño”. Reference: 19490/PI/14.

ANSYS Inc. 2010. ANSYS CFX. Solver Theory Guide. Release 13.0.

Castillo, L. & Carrillo, J.M. 2012. Numerical simulation and validation of intake systems with CFD methodology. Proc. 2nd IAHR European Congress; Munich, 27–29 June 2012. Castillo, L.G., Carrillo, J.M. & García, J.T. 2013a. Comparison of clear water flow and sediment flow through bottom racks using some lab measurements and CFD methodology. Proc. Seven River Basin Management. Wessex Institute of Technology; New Forest, 22–24 May 2013. Castillo, L.G., Carrillo, J.M. & García, J.T. 2013b. Flow and sediment transport through bottom racks. CFD application and verification with experimental measurements. Proc. 35th IAHR Congress, Chengdu, 8–13 September 2013. Castillo, L.G:, García, J.T., and Carrillo, J.M. 2014. Experimental measurements of flow and sediment transport through bottom racks. Influence of graves sizes on the rack, Proc. International Conference on Fluvial Hydraulics, 2014. Castillo, L.G:, García, J.T., and Carrillo, J.M. 2015. Effective void ratio and rack length definition by experimental measurements of flow with gravel size sediments through bottom racks. E-proceedings of the 36th IAHR World Congress 28 June – 3 July, 2015, The Hague, the Netherlands. Castillo, L.G:, García, J.T., and Carrillo, J.M. 2016. “Experimental and Numerical Study of Bottom Rack Occlusion by Flow with Gravel-Sized Sediment Application to Ephemeral Streams in Semi-Arid Regions”.

Water 2016, 8, 166; doi:10.3390/w8040166. Castro-Orgaz, O., W. H. Hager. 2011. “Spatially-varied open channel flow equations with vertical inertia”. Journal of Hydraulic Research, 49:5, 667–675. De Marchi, G. 1947. “Profili longitudinali della superficie libera delle correnti permanenti lineari con portata progressivamente crescente o progressivamente decrescente entro canali di sezione costante”. Ricerca scientifica e ricostruzione. Garot, F. 1939. De Watervang met liggend rooster, De Ingenieur in Nederlandsch Indie, 1939. Mostkow, M. 1957. Sur le calcul des grilles de prise d’eau, La Houille Blanche, 4, 569–576, 1957. Nakagawa, H. 1969. On Hydraulic performance of bottom diversión Works. Bulletin of Disaster Prevention Research Institute, Kyoto University. Nasser, M. S., P. Venkataraman & A. S. Ramamurthy 1980. Flow in a channel with a slot in the bed, Journal of Hydraulic Research, 18:4, 359–367. Noseda, G. 1956. Correnti permanenti con portata progressivamente decrescente, defluenti su griglie di fondo, L’Energia Elettrica, 565–581, 1956. Orth, J., Chardonnet, E. & Meynardi, G. 1954. Étude de grilles pour prises d’eau du type. La Houille Blanche 3: 343–351. Righetti, M., and Lanzoni, S. 2008. Experimental Study of the Flow Field over Bottom Intake Racks. Journal of Hydraulic Engineering 134(1): 15–22. Thielicke, W. and Stamhuis, E. J. 2014. PIVlab – TimeResolved Digital Particle Image Velocimetry Tool for MATLAB (version: 1.4). http://dx.doi.org/10.6084/m9. figshare.1092508. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Image processing techniques for velocity estimation in highly aerated flows:

Bubble Image Velocimetry vs. Optical Flow

D.B. Bung & D. Valero

Hydraulic Engineering Section, FH Aachen University of Applied Sciences, Aachen, Germany

ABSTRACT: Measuring of flow velocities in aerated flows is known to be difficult in physical models. Appli

cation of classical anemometers or ADV probes is limited to low air concentration. Thus, highly aerated flows

are commonly investigated by use of intrusive needle probes (conductivity or optical fiber) which allow deter

mination of both, air concentration and velocity, as well
as related parameters (e.g. bubble chord lengths and

turbulence). In the recent past, non-intrusive image
processing techniques have gained more attraction. In the

present paper, Bubble Image Velocimetry and Optical Flow
methods are applied to aerated stepped spillway

flows using high-speed cameras with different resolution to
highlight capabilities and limitations of both meth

ods. Results show that Optical Flow is capable to give
results of at least the same accuracy as Bubble Image

Velocimetry. A higher image resolution enhances the quality
of the results. The dense velocity information being

obtained by Optical Flow may help to carry out
investigations on turbulence in future.

1 INTRODUCTION

In hydraulic engineering, the hydraulic performance of

weirs and spillways is commonly elaborated by means

of hydraulic modeling during early design stages.

Herein, energy dissipation potential is a key feature

which is closely linked to the flow velocity and fric

tion factor. Performing of velocity measurements is

mostly difficult due to self-aeration of the water. Clas

sical velocity probes, e.g. anemometers or Acoustic

Doppler Velocimeters (ADV) become less accurate or

even unsuitable in highly aerated flows (Matos et al.

2002). Alternatively, intrusive conductivity or fiber

optical probes are commonly employed (Boes 2000,

Bung 2011a, Chanson 2002, Felder & Chanson 2015).

However, these probes are known to be expensive and very sensitive to damages due to very fine tips.

In the recent past, non-intrusive imaging techniques have been adopted to such highly aerated spillway flows in several laboratory studies. The advantage of these imaging studies is given by the flow velocity fields which are directly obtained. In contrary, intru sive needle probes yield time-averaged, local velocity (and air concentration) data only. Most of these non intrusive studies were based on the so-called Bubble Image Velocimetry technique (BIV) which was intro duced by Ryu et al. (2005) for impinging waves. BIV is the application of a common Particle Image Velocime try (PIV) method to bubbly flows using bubbles as tracers. Bung (2011b) first demonstrated the appli cability of this technique to stepped spillway flows with high air concentrations. These results were later improved by Leandro et al. (2014) applying a complex calibration method. Bung &Valero (2015) showed that some image pre-processing and filtering may also help to improve the results compared to data from a conductivity probe. Anyway, all studies showed that BIV tends to underestimate the conductivity probe data in order of up to 20 %. Another imaging technique that has recently been applied to highly aerated flows (Bung & Valero 2016a) as well as to slightly aerated flows and particle images for PIV (Bung & Valero 2016b) is given by the socalled Optical Flow (OF). This method is known in the Computer Vision community and is applied to different fields like autonomous car-driving or video compression (Fortun et al. 2015). The optical flow is defined as the movement of brightness (intensity) patterns through a sequence of images (Horn & Schunck 1981). It is assumed that the brightness of a moving pixel is conserved through a set of images. While the Horn & Schunck method is a global method, local methods (e.g. Lucas & Kanade 1981), consider a constant optical flow within a close neighborhood. Thus, local methods do not allow to determine a movement within a region of uniform intensity leading to sparse velocity fields. Global methods, in contrary, yield dense velocity fields with velocity information at every image pixel. Although several improvements of the classical Horn & Schunck method have been presented since its original formulation, Bung & Valero (2016a, 2016b) showed that the original method gives good results when applied to hydraulic engineering problems. Similar results were identified by Corpetti et al. (2006), Liu et al. (2015) and Liu & Shen (2008). The present paper aims to compare the results from BIV with those from OF for two different image resolutions and quantify and discuss their differences.

Figure 1. Extracted high-speed images from both setups,

flow direction from left to right.

2 METHODOLOGY

Both imaging techniques are applied to high-speed

video frames which are taken at two different stepped

spillway models. Both models are set up with 1V:2H

slope, 6 cm step height and similar inflow conditions.

The flumes are built with Plexiglas sidewalls to allow

visual inspection and with a black PVC rear wall to

enhance the contrast. The specific discharge is set to

0.07 m^2/s.

The first experimental setup consists of a 30 cm

wide flume with a total drop height of 2.34 m. Videos

are captured with a KSV instruments high-speed cam

era, type HiSiS 2000, with a resolution of 256 × 256 px

and a sample rate of 1220 fps.The camera is installed at

50 cm distance of the sidewall and the flow field is illu

minated with halogen spot lights with a total power of 1,300 W (installed above the flume and in front of the Plexiglas sidewall). Discharge is controlled with a flap valve and an inductive flow meter. Measurements are limited to step 28 in the fully-developed flow region. Figure 1a shows an exemplary frame. Please note that this frame is cropped to the field of interest and rotated to align the pseudo-bottom formed by the step edges with the horizontal. The pixel density is ~14 px/cm. The reader may note the strong blurring of the image. The second installation consists of a 50 cm wide flume with a total drop height of 1.74 m. Videos are recorded with MotionScope Miro M120 camera provided by LaVision with a resolution of 1920 × 1200 px and a sample rate of 732 Hz. The camera is installed at 60 cm of the sidewall. The video is captured at step 21 in the quasi-uniform flow region (Bung & Valero 2016a) using a synchronized LED illumination installed in front of the flume. The discharge is controlled with use of a frequency-regulated pump and an inductive flow meter. Figure 1b shows an extracted video frame (again rotated and cropped to the field of interest). The pixel density is here ~103 px/cm. Determination of velocity fields is conducted using MatPIV (Sveen 2004) in its latest version 1.6.1 for BIV. MatPIV is an open-source Matlab ® toolbox allowing to set several filters like a signal-to-noise ratio filter (SNR) as well as local and global filters to be applied with user-defined thresholds. For OF, another open-source Matlab ® toolbox, developed by Sun et al. (2010), is used. This toolbox was written to benchmark recent advances in optical flow methods against the classical Horn & Schunck scheme which was found to be still competitive when appropriately implemented.This classical Horn & Schunck method, which will be described in the next section, is used in the present study and combined with an image processing technique to improve the results. All results will be compared to conductivity probe data which were obtained in the centerline of setup 1 (Bung 2009). It is acknowledged that wall effects are very likely, but assumed to be small (cmp. Leandro et al. 2014). Another drawback of these imaging techniques is that no clear measuring plane can be defined as no laser sheet is applied as in standard PIV setups. The reader may note that application of such laser systems is not possible in highly aerated flows due to strong reflections and uncontrolled redirection of the laser light. 3 IMAGING TECHNIQUES 3.1 Bubble Image Velocimetry Displacements in Bubble Image Velocimetry (BIV) or Particle Image Velocimetry (PIV), respectively, are generally determined by performing a twodimensional cross-correlation analysis of the image intensity matrices of two subsequent frames:

In Eq, (1), I_1 and I_2 denote the pixel intensities of two images, X and Y the pixel coordinates and Δx and Δy the increment to the second frame. This cross correlation function is characterized by a peak value which represents the best correlation and thus, the most likely displacement U and V in X and Y direction. In order to obtain a complete displacement field, images are commonly divided into multiple interrogation win dows, which may also overlap, and Eq. (1) is solved for each window.

Speed-up of the cross-correlation may be achieved when transferring the images into their frequency domain (i.e.: Fast Fourier Transformation or FFT). According to the correlation theorem, correlation in the spatial domain is equivalent to multiplication in the frequency domain. An inverse FFT then yields the spatial cross-correlation.

In any case, it is pointed out that for each interroga tion window, a single displacement vector is obtained. Denser displacement or velocity fields thus require a high number of interrogation windows whereby the minimum window size is limited as particles must not leave the window from one image to the next. Additionally, when applying a FFT cross-correlation, displacements larger then half of the window size lead to aliasing, i.e. the correlation peak is folded back into the correlation plane but appears on the opposite side (Sveen 2004). Alternatively, higher sample rates could be chosen (technically limited) to allow for reducing interrogation window sizes and obtain denser velocity fields.

3.2 Optical Flow

In order to determine the movement of brightness pat terns, or the movement of an object, respectively, the classical Horn & Schunck method basically assumes the conservation of brightness $I(X, Y, t)$ of a mov ing pixel with the image coordinates X and Y at the time t:

Equation (1) may be expanded to

or be written in a compact form as

where I_X and I_Y are the spatial image brightness deriva tives in X and Y direction and I_t is the temporal derivative. U and V are the unknown spatial displace ments in X and Y direction, respectively. Equation (4) is an ill-posed problem with 2 unknowns, requiring a second constraint to be solved.

Horn & Schunck (1981) assumed that neighbour

ing points have a similar velocity.A so-called “gradient constraint equation” or “data term” was proposed by reducing the square of the gradients of the optical flow velocity. With this assumption, Eq. (4) can be substituted by the following objective function where E is the error functional to minimize, $\nabla = (\partial/\partial X , \partial/\partial Y)$ is the spatial gradient and α a scaling factor for the spatial term. Larger values for α will result in smaller flow gradients and thus smoother flow fields. Additional image filtering techniques are known to improve the results. As shown by Bung & Valero (2016a, 2016b), incremental multi-resolution techniques based on the so-called pyramid method help to enhance the results when OF is applied to aerated flows. In principle, this method applies a single filter to a set of images instead of using different filters to a single image (Adelson et al. 1984, Burt & Adelson 1983). For this purpose, the original image size is reduced resulting in a blurred (low-pass filtered) copy of the image when again upscaled to its original size. When this step is repeated multiple times, a set of images with different sizes is obtained forming a pyramid when illustrated one above the other.This pyramid may be regarded as the spatial-frequency domain of the image. The image pyramid method then downsamples each level from its nearest finer level and the optical flow is first estimated at a coarse level. The coarse scale displacement is used afterwards to correct the sequence at the next finer level (warping) (Sun et al. 2010).The total displacement is then given by the summation of all motion increments. According to Bruhn et al. (2005), this method results in higher accuracy for flow with large displacements. Sun et al. (2010) state that there is no optimal method which is suitable for any arbitrary sequence. Instead, a suitable method needs to be chosen for a given sequence. The reader is referred to the review papers of e.g. Fortun et al. (2015) or Sun et al. (2010) that give further descriptions of optical flow techniques. 4 RESULTS 4.1 Low-resolution frames Velocity fields from the low-resolution images have been presented by Leandro et al. (2014) applying a complex calibration scheme with variation of different filter settings and interrogation window sizes. In order to obtain smoother velocity fields, multiple BIV calculations were performed using a series of subsequent images and the median velocity field was assumed to be representative (cmp. Bung 2011b). It was found that the best results were obtained when using 800 frames in total and an interrogation window size of 16 × 16 px with 75% overlap (the displacement

Figure 2. BIV velocity magnitude field (in cm/s) after cal

ibra-tion according to Leandro et al. (2014), median result

from 800 frame pairs (note that the vectors have been nor malized to bet-ter visualize the step niche vortex, flow from

left to right).

Figure 3. Velocity profiles at upstream step edge ($x = 0$ cm) from OF calculations with 1, 10, 50, 200 and 800 frame pairs compared to the BIV results from Fig. 2 and the conductivity probe data (CP) for the low-resolution frames.

between two frames is about 5 px). In the current study, the settings from this calibration are adopted leading to the BIV velocity field in Fig. 2.

Fig. 3 compares the BIV velocity profile which was extracted at the upstream step edge with the results from OF. For this purpose, OF results will be also averaged over a varied number of frame pairs. In order to evaluate the required number of frame pairs, median velocities from 10, 50, 200 and 800 OF cal culations will be compared. The conductivity probe data is included for completeness. The image pyramid technique is applied with 4 levels and an image size reduction of 50% between each level. The minimum image size is thus 16×16 px. Smaller image sizes are known to not significantly improve the results (Bung and Valero, 2016a; Sun et al., 2010). A maximum of 10 iterations is set for each level and pixel to solve Eq. (5).

Figure 3 shows that BIV and OF give similar results when median data from the same number of Figure 4. Median OF velocity magnitude field (in cm/s) using 200 low-resolution frame pairs (note that the vectors have been normalized to better visualize the step niche vortex, every 5th vector is displayed for better legibility, flow from left to right). frame pairs is taken into account. OF results converge after roughly 200 frame pairs. It is noted that OF gives higher velocities with only a single frame pair which are then closer to the conductivity probe data. However, significant scattering is found. For more frame pairs, velocities increase up to ~200 cm/s and compare well with BIV. Both methods significantly underestimate the CP data. The complete velocity field from OF using 200 frame pairs is presented in Fig. 4. As expected, this velocity field appears much smoother than the BIV result in Fig. 2. While the stagnation point on the horizontal step surface is found at the same position, the core of the step niche vortex is slightly shifted downstream. Another difference is found at the shear region at the pseudo-bottom ($z = 0$) which is again smoother compared to the BIV results although the median is determined using much less frames in OF. 4.2 High-resolution frames In order to evaluate the sensitivity of velocity fields to initial image quality, similar BIV and OF calculations are performed with images from setup 2. BIV is carried out with an interrogation window size of 80 × 80 px (the maximum pixel displacement is ~40 px) with 75% overlap. OF is performed with 7 image pyramid levels taking into account the median of 5, 20, 50 and 100 frame pairs while BIV is again limited to the maximum frame number. Figure 5 presents the extracted velocity profiles from both imaging techniques. It is found that BIV and OF results compare fairly well for the same number of frames. OF tends to give some higher flow velocities, particularly in lower flow elevations and thus, more accurate results. Interestingly, a decrease of median flow velocity is found for increasing number of frames. This trend was not clearly apparent in Fig. 3. Figure 6 presents the complete velocity field from OF using 100 frames. Both, Figs. 5 and 6a, show the high density of data being obtained by the OF method.

Figure 5. Velocity profiles at upstream step edge ($x = 0$ cm)

from OF calculations with 1, 5, 20, 50 and 100 frame pairs

compared to the BIV results with 100 frame pairs and the

conductivity probe data (CP) for the high-resolution frames.

Figure 6. Median velocity magnitude fields (in cm/s) using

100 high-resolution frame pairs (note that the vectors have

been normalized to better visualize the step niche vortex, flow

from left to right).

Figure 6b illustrates the BIV results. Apparently, OF

and BIV results compare fairly well again.

5 DISCUSSION & OUTLOOK

It was shown that the OF method is capable to give

velocity data with the same or even higher accuracy

as the BIV method. It must be noted that PIV is a statistical and integral approach which is generally better suitable to finite particles. As stated by Liu et al. (2015), OF, as a differential approach in contrary, is better suitable to images with continuous patterns which are also found in aerated flows. However, in higher elevations where the air concentration is very high, homogenous pixel intensity with low intensity gradients and strong blurring is typically found, yielding a high deviation to the conductivity probe data which was recorded at the channel centreline. Additionally, wall effects and the known characteristic of intrusive needle probe to overestimate flow velocities may also enhance this deviation. A noticeable advantage of the applied Horn & Schunck method is given by the high-density data. Calculation time is consequently higher than for BIV with the same frame number when highresolution frames are chosen (by a factor of 50). For low-resolution frames, calculation effort increase is almost negligible. However, the authors believe that this high data density could be useful for determination of small-scale turbulence structures. Therefore, future work is intended to better understand the sensitivity to different parameter settings. Some exemplary results on velocity fluctuations u fluct , v fluct and turbulence intensity Tu defined by based on the flow velocities u in x-direction and v in y-direction, respectively, are given in Fig. 7 showing the related profiles at the upstream step edge ($x = 0$ cm). The results are compared to experimental data for Tu by Amador et al. (2006) obtained with PIV in the non-aerated flow of a stepped spillway with 1V:0.8H, 5 cm step height and 0.11 m

2 /s specific discharge. Experimental data is limited to the boundary layer region. The elevation is normalized by h 90 , i.e. the water level with an air concentration of 90%. Apparently, the shape of the turbulence intensity distribution compares fairly well with the experimental data. Higher turbulence intensities in the current study may be explained by the high aeration and the airwater interaction. It is also observed that the vertical velocity fluctuations v fluct are much more intense than u fluct . However, the reader may note that the absolute fluctuations are of one magnitude smaller than those in x-direction (not illustrated). The complete velocity fluctuation field in x-direction is presented in Fig. 8. Significant turbulence is found within the shear layer developing downstream of the step edge.

Figure 7. Velocity fluctuations u fluct and v fluct and tur

bulence intensity Tu from OF calculation with 100

high-resolution frames (lines) and Amador et al. (2006) for

non-aerated region (markers).

Figure 8. Velocity fluctuation u fluct (in %) in x-direction

according to Eq. (6) from OF calculation with 100

high-resolution frames.

6 SUMMARY & CONCLUSIONS

A comparative study of Bubble Image Velocimetry

and Optical Flow for non-intrusive velocity determi

nation in highly aerated stepped spillway flows has

been conducted. It was found that both methods give

similar results while OF tends to be more accurate

than BIV. However, the deviation of both imaging

methods compared to intrusive measurements with a

conductivity probe is still significant. Further stud

ies using different OF schemes will be conducted in future to analyze potential improvements. It was found that some improvement may be obtained using high resolution images. However, OF requires much more calculation time which may be reasonable as a dense velocity field is obtained when a global method, such as the Horn & Schunck method, is employed. It was shown that this high data density may be useful to investigate small-scale structures. When a high sample

Lucas, B.D. & Kanade, T. 1981. An iterative image registration technique with an application to stereo vision. Proceedings of imaging understanding workshop, 121–130.

Matos, J., Frizell, K.H.,André, S. & Frizell, K.W. 2002. On the performance of velocity measurement techniques in air water flows. Hydraulic Measurements and Experimental Methods, 1–11. Ryu,Y., Chang, K.-A. & Lim, H.-J. 2005. Use of bubble image velocimetry for measurement of plunging wave impinging on structure and associated greenwater, Meas. Sci. Technol., 16 (10), 1945–1953. Sveen, J.K. 2004. An introduction to MatPIV v.1.6.1, Department of Mathematics, Mechanics and applied Mathematics, University of Oslo. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Biomechanical tests of aquatic plant stems: Techniques and methodology

A.M. Łoboda, Ł. Przyborowski & M. Karpiński

Institute of Geophysics, Polish Academy of Sciences, Warsaw, Poland

R.J. Bialik

Institute of Geophysics, Polish Academy of Sciences, Warsaw, Poland

Institute of Biochemistry and Biophysics, Polish Academy of Sciences, Warsaw, Poland

ABSTRACT: Six different aquatic plant species (Myriophyllum spicatum L., Potamogeton crispus L., Pota

mogeton pectinatus L., Ceratophyllum demersum L., Callitriche palustris L. and Elodea canadenis L.) were

collected from two lowland, sandy-bed rivers (the Wilga and the S´wider), in order to conduct a series of

biomechanical tests (three-point bending and tension). The tests were carried out to improve knowledge of

the biomechanical properties of aquatic plant stems and to show problems with the techniques and methodology

of such tests which have not been pointed out in previous works. These tests were made in dry and wet conditions

with a benchtop materials testing machine. The differences between results obtained in different conditions are

presented. Significant problems, such as choosing the right way to protect the ends of the sample from damage in

the clamps of the device used and minimizing the time after removal of the sample from water for measurement,

are discussed briefly. The diameter and cross-section of the stem, sample and gauge lengths were measured for

each case. The conducted trial tests show that the diameter of the stem, its internal structure and the growth

stages of plants are of great significance regarding the value of the measured forces. The obtained results could

be crucial in identifying a relationship between plant biomechanics and flow resistance.

Keywords: aquatic plants, bending test, tension test, biomechanics

1 INTRODUCTION

Studies of plant systems on different scales are con ducted with the use of different mechanical tools (Moulia 2013). Because of the large number of species of aquatic plants, which are characterized by different properties, our knowledge in this field is still incomplete. Many researchers, including Niklas (1992), Boudaoud (2010), Jordan and Dumais (2010), Wojtaszek (2011) and Ennos (2011), have carried out experiments with macrophytes to study plant biome chanical properties. Moreover, studying biomechanics provides new insights for other sciences, like biol ogy, medicine or engineering. In addition, progress in understanding fluid mechanics has resulted in an increasing number of researches, e.g. Nikora (2010), Koehl et al. (2008), Puijalon et al. (2011), seeking answers to the question of how aquatic plants inter act with waves and water streams (Moulia 2013). For example, Madsen et al. (2001) noted that the drag force measured on plants with the same structure, size and species was not constant, and thus biomechani cal properties of aquatic plants are difficult to study because of their complexity.

It is important to note that imposing mechanical

stress on plants from the field can lead to very different results from tests conducted on laboratory-growth plants of the same species, as suggested by Schutten et al. (2005).

Therefore, minimizing errors by searching for optimal experimental procedures and testing plants from the field should benefit future investigations aimed at understanding plant biomechanics. In addition, fluvial hydraulics provides quantitative data which extend the understanding of fluvial phenomena. Scientific projects on fluvial hydraulics are usually carried out in controlled laboratory settings, but should be validated through field measurements, systematic field studies and field experiments (Sukhodolov 2015). In recent years, extensive studies have been carried out on vegetated flows. This generally involves the exploration of flow-plant interactions on multiple scales, as well as plant reconfigurations and the resulting changes to flow drag (Sukhodolov 2015). To improve our knowledge about hydrodynamic processes occurring in streams, the study of biomechanical properties of aquatic plants is required. Vegetation found in the river bed strongly affects the conditions of the dominant flow of water. An important result of higher bottom sedimentation is change in the velocity field and increasing flow resistance. On the other hand, the significance of aquatic vegetation has been widely acknowledged. Aquatic

plants contribute to the biodiversity of water envi

ronments and create habitat for numerous animal

species. Fluvial vegetation also influences the qual

ity of water through nutrient uptake and increased

oxygen production (Nepf 2012, Siniscalchi & Nikora

2013). Furthermore, water plants play a key role in

reducing coastal erosion (Nepf 2012). Most recently,

Sukhodolov (2015) recognized the potential appli

cation of relevant research results in the emerging

branch of recreational hydraulics. The distribution of

water plants in various environmental conditions, most

importantly varying flow velocities and the resultant

drag forces, is determined by species-specific mor phological and biomechanical properties. The plant’s adaptational ability to reduce water flow-induced drag forces exerted on it are crucial for its survival (Miler et al. 2012). Biomechanical features (i.e., bending and tensile properties) affect the plant reconfigura tion in streams. A significant amount of research on biomechanical properties of plants focuses on terres trial, rather than aquatic, plants (Miler et al. 2010, 2012, 2014, Niklas 1992). Current research on biome chanical properties of aquatic vegetation is generally aimed at discovering local adaptation mechanisms through investigating individual plant species. Broadly explored biomechanical properties include the charac teristics of plant stem bending and tension resistance as well as leaf shape, flexural rigidity, serration and roughness (Bocia¸g et al. 2009, Kałuz˙a & Tymin´ski 2010, Albayrak et al. 2012, Miler et al. 2012).

At the plant stem scale, Bocia¸g et al. (2009) car ried out experiments on stem strength in submerged macrophytes. The bending and tensile properties of four species were measured. Measurements were col lected in field conditions from three lakes and three rivers in Poland in July. The comparison of mechani cal properties of plants was based on tensile strength, which was obtained from stretching tests. In addition, Bocia̧g et al. (2009) concluded that stretching, twisting and bending of stems is cause by hydrodynamic forces, and thus the survival of an aquatic species depends on its resistance to bending and tension. Aquatic plants have better adaptation to river habitat in flowing rather than in stagnant water, because in flowing water they increase their tensile resistance. In this study it was demonstrated that species of macrophytes which have tension resistance also have a larger cross-sectional area and occur in flowing water. It was suggested that changes in biomass allocation, e.g., cell wall thicken ing, might be an adaptation response to mechanical forces. It was also suggested that these properties change for each individual specimen because they come as a response to particular environments and not as a genetic property (Bocia̧g et al. 2009).

Subsequent research similar to that of Bocia̧g et al. (2009) examining biomechanics of aquatic plants was performed by Miler et al. (2012). For four submerged macrophytes they examined properties such as ten sion, bending and elasticity. The aim of the study was to establish the relationship between biomechan ical properties of plants and their habitats. The plants selected for the study were different in terms of morphology and biomechanical characteristics. The flow drag and the interaction between aquatic species and their habitat have become a frequent topic in hydraulic studies. Due to the complexity of the phenomenon, the research into individual plant species provides important insights into adaptation mechanisms.To discover the bonds between biomechanics of aquatic plants and turbulent flow around them one has to study the biomechanical properties first. As for the Manning's coefficient in order to acquire proper value the banks and bed properties have to be known. Thus, to improve our knowledge about the impact of flow characteristics around plants information about the stiffness and ability to motion of plants is necessary. The changes of shape and frontal area of the plant alter the shear boundary layer around the stem, therefore they have impact on eddies generation due to the drag force. This paper shows the issues and problems that were encountered during testing of the biomechanical properties of aquatic plants. A new approach was also implemented to check how the test conditions influence the obtained results.To the best of our knowledge, most if not all previous studies carried out biomechanical tests in dry conditions. It is not clear enough how the tests were performed, as some important aspects were omitted. For example, the time of preparing specimens or their humidity could vary the final values of force, strain, stress, etc. Therefore, the main goal of this paper is to share our experience of conducting the described biomechanical tests. It should help by allowing better conclusions to be inferred from the results obtained in both past and future studies. 2 METHODOLOGY 2.1 Sampling sites The plants were collected from two sites, first on 26 August 2015 from the River Wilga and second on 8 September 2015 from the River S´wider. These are right tributaries of the Vistula, located, about 50 and 10 km south of Warsaw, respectively. In collecting places, both rivers had unregulated banks, and the river bed was sandy. Callitriche palustris L., Potamogeton pectinatus L. and Elodea canadenis L. species were collected in the Wilga, and in the S´wider Potamogeton crispus L., Potamogeton pectinatus L., Ceratophyllum demersum L. and Myriophyllum spicatum L. were gathered. Physical conditions of both sites were similar; water velocity was about 0.3 m/s, and depth did not exceed 0.6 m. On the Wilga, the sampling site was full of gravel and small stones. Potamogeton pecinatus L. and Callitriche palustris L. grow there in abundance while on the Wilga plants were scarce and were found only near the river banks in small groups. Plants with roots were gently removed from the bottom with a spade and were transported immediately in two plastic aquariums specially prepared for this purpose, which were equipped with an aeration device.

2.2 Plant species and equipment

Aquatic plants collected in the S´wider and the Wilga rivers include six perennial herb species common in Polish lowland rivers. They represent a wide spectrum of morphological variability, from short, thin and del icate, to long, durable and heavy-branched species. The analysed species are as follows: Potamogeton crispus L., Myriophyllum spicatum L., Ceratophyllum demersum L., Potamogeton pectinatus L., Callitriche palustris L., and Elodea canadenis L. Figure 1 shows pictures of the first four of them, counting from left to right, with the bar representing length of 50 mm. Dur ing the first week of the experiment, when Callitriche palustris L. and Elodea canadenis L. were tested, no pictures were taken.

The drought occurred during the last few weeks of the summer. It was assumed that a large number of plants had been damaged or died because of this hydrological phenomenon. The previously identified canopies of different species were much smaller and in the case of Potamogeton crispus L. it was found in the early stage of growth. This species had characteristic curly leaves and stiff stems with many branches, and its mean diameter was 1.6 mm. Potamogeton pecinatus L. was very similar, except it was more branched, leaves were straight, and the mean diameter of the stem was thinner: from the Wilga it was 1.2 mm and from the Świder 0.9 mm. Because plants were shallow-rooted in sand, some roots were also tested; their diameter was about 2 mm. This species occurred in abundance in the Wilga river. Myriophillum spicatum L. had the longest, most rigid, reddish stem and its mean diameter was about 2.1 mm. It was rarely branched and leaves were short and delicate. Also, this species had a unique wheel-like internal structure whereas all the other species had a honeycomb structure. Ceratophyllum demersum L. was a delicate species, found in

Figure 1. Photograph of four tested specimens, from the left to right: Potamogeton crispus L., Myriophyllum spicatum L.,

Ceratophyllum demersum L., Potamogeton pecinatus L. The bar has length of 50 mm. two places in the Świder river. Its stem was mediumbranched compared with other plants, and its mean diameter was very small at only 1.06 mm. Callitriche palustris L. was a very delicate plant, but it had over a dozen stems from one shoot that were very tangled. Single stems had a diameter below 1 mm and were too limp for fixing to the bench top testing machine. Elodea canadeis L. was probably harmed by the drought, because specimens had short, fresh shoots, growing from ligneous root parts. Its stems were stiff and its mean diameter was 1 mm. Laboratory equipment was tailored to the needs of the experiment. It comprised a Tinius Olsen Benchtop Materials Testing Machine 5ST with 500 N load cell, a 112-litre aquarium with necessary devices to sustain appropriate conditions for stored plants and a biological microscope for diameter measurements and observing internal stem structure. The machine was additionally equipped with stainless steel scaffolding to allow submerging of plants in water during tests. Figure 2 shows a schematic drawing of the machine and its set-up for bending and tension tests. A stainless steel bucket was placed on the machine, and the scaffolding reached the bucket's bottom so a lower clamp or support bars could be installed underwater. The bucket was deep enough to cover the bending head or upper clamp with water but only when sample length did not exceed 10 cm. The machine had a maximal frequency of force sampling equal to 1000 samples per second, and the measurement error was no higher than 0.5% and displacement accuracy was 0.0001 mm. Horizon software was used to operate the machine and export results to Matlab. From each conducted test a maximum of 10,000 records could be obtained. The aquarium was equipped with two pumps (Aquael 2Circulator 500) that imitated water flow, a standard aeration device and two fluorescent light bulbs (Yuwel T5 HiLite 2 × 28W) to simulate natural sunlight for

Figure 2. Schematic drawing of the bench top testing machine – a) front view, b) bending test setting, A – distance between

support bars, c) tension test setting, B – distance between clamps.

12 hours per day. Additionally, each day, liquid CO_2

(glutaraldehyde) was added to stimulate plant growth.

Plants were rooted in a 5 cm thick layer of fine gravel.

A biological microscope (Delta optical Genetic Pro

Trino) was also equipped with a 5 MP microscopic

camera (DLT-CAM PRO 5MP USB 2.0) for taking

pictures and to measure diameter with the help of a

0.01 mm scale glass slide.

2.3 Preparation of specimens

After plants were transported to the laboratory, they

were placed into a glass tank with fresh water, pumps,

aerator, and light provided to make the conditions as

natural as possible. Plants were attached to a bottom of

the tank filled with fine gravel. Each set of tests was

conducted within a week of the delivery of submerged macrophytes to the laboratory. It prevented the situa tion whereby plants could change their characteristics in an attempt to adjust to a different environment. Dur ing the time of the tests, each single stem was taken up from the tank and cut into pieces 5 to 8 cm long. The length depended on the type of test. Then prepared pieces, up to 10 in a row, were immediately measured with a microscope. For the tension test, after measure ment, the ends of the pieces were glued to short strips of sandpaper. The glue was cyanoacrylate (superglue), the sandpaper was thin and its granulation was fine. A strip of sandpaper was bent in the middle to wrap the stem end from three sides. After wrapping, glue was Figure 3. Potamogeton crispus L. stem glued to sandpaper ‘leaves’. placed inside between the stem and the grit side of the paper. That produced at each end of each stem short (about 1 cm in length and 0.5 cm in width) “leaves” made of sandpaper, which enabled us to prevent stems from tearing in the machine clamps and ensured easier fixing. Figure 3 shows a prepared piece of stem. 2.4 Tension tests and three-point bending tests Tension tests give information about how plant stem can be prolonged. The prolongation increases the area of the plant which has impact on the drag force.

The drag force causes turbulence along the plant. On the other hand, bending tests demonstrate how much the plant is prone to bend. The plant, which can be eas ily bend, produces more turbulent water motions than the plant with a stiff stem.

Tension tests were performed within 50 minutes from the moment when plants were taken out of the aquarium to the moment when the last of the prepared pieces in a row was tested in the bench top testing machine. Each prepared stem piece was fixed between machine clamps in a vertical position, whereby the upper clamp was pulling up during testing. Each time, measurement of the distance between clamp points was necessary to calculate strain during testing. Pre cise calibration of clamp height was needed in some cases, when the fixed stem was not tight enough. The rate of displacement was 10 mm per minute. Each test ended when stem break occurred. Horizon, the soft ware that was used to operate the bench top testing machine, recorded force, time and displacement, cal culating further values such as strain and stress when it was given data describing current gauge length and stem diameter.

The bending test required plant pieces cut to a length of 68 mm or 72 mm to match the gap between sup port bars, where stem pieces were laid. The gap had a distance 46 mm or 49 mm, depending on the plant species. The point of bending lies exactly at the half of distance between support bars. Such a setting allowed us to perform three-point bending tests. Bending tests were performed only for Potamogeton pecinatus L. and Myriophillum spicatum L. The rest of the plants were too limp or too prone to slip easily which meant that the force could not be measured. The rate of lowering a bending head was 20 mm per minute and the end ing condition was displacement of a bending head by 2.5 cm, which mostly caused the stem to be bent at an angle greater than 45 degrees.

3 RESULTS

The preliminary results showed a slight difference in breaking force and strain values between the experi ments conducted in dry and in wet conditions. More detailed results with characteristics like Young’s mod ulus will be calculated in future, when more specimens will be tested by the same methodology. The unit of force was the Newton, and displacement of bending bar was in millimetres. The strain was a magnitude of change in the length which occurs when a force is applied. It was calculated by the following equation (Niklas 1992):

where l 0 is original length of the measured dimen sion and l is length after application of force in the tension test. In the bending test l 0 is a starting posi tion of bending head and l is a displacement of it.

The break strain values and the break force values were
Figure 4. Force-strain curve for Potamogeton pecinatus L. in tension test. found manually for each specimen from the recorded data, instead of depending on Horizon software calculations, because Horizon calculates the break point

when it meets a certain percentage difference in the following force values, but actually most of such drops were not connected with specimen break. The drops and leaps in the forcestrain curve were caused by: straightening curved stem sections, fitting the sand paper “leaves” between the clamps (its surface had small metal bumps), and the accuracy of the machine measurement that even without any load showed force oscillating around 0.001 N. As real values of break force and break strain, the values of force after reaching the top level and before dropping down to almost or below zero value were taken. When the corresponding strain values were almost the same for a few records, maximal value of the force for a given percentage of strain was taken, because it was the point when the first crack in a stem had probably occurred, and slowly dropping force values meant widening of rupture through the periphery. That caused overall break of the stem, while the strain value stayed the same. Figure 4 shows the typical force-strain curve, where the breaking point read by Horizon and the actual breaking point are visible. 3.1 Tension tests Potamogeton pecinatus L., of which over 200 pieces were tested, was found to be the most resistant to stretching. The ratio of used force to acquired extension of stem piece was highest among studied plants. Also, the force required to break stems was the highest (almost 2.5 N) but the strain was similar to other species. The dry tests showed that break force and break strain are both lower than in tests performed in water. Ceratophyllum demersum L., a plant with mean diameter of a stem very similar to Potamogeton pecinatus L., was much more fragile than latter one. The break force was about five times lower and the break strain was about a half lower than these values for Potamogeton pecinatus L..The differences between dry and wet tests had again showed that the tests conducted in water gave higher results. The most elastic of the tested plants was Myriophillum spicatum L.; its maximal strain was above 10 % while used force did not

Table 1. Comparison of mean values in tension test of

Potamogeton pecinatus L. and Potamogeton crispus L. Species

Parameter	Unit	P. pecinatus L.	P. crispus L.
Mean force in dry tests	N	2.52	1.65
Mean force in wet tests	N	2.70	1.39
Mean strain in dry tests	%	7.90	7.70

Mean strain in wet tests % 9.19 6.75

exceed 2 N. In wet tests the maximal strain was slightly lower than in dry tests and the break force comparison showed the opposite. Altogether, only 50 tests were performed for that species, so it was hard to compare dry and wet tension tests. Myriophillum spicatum L's. unusually high rate of strain was the effect of a wheel like internal structure (Miler et al. 2012) which did not occur in other studied plants. Potamogeton crispus L. was very similar in maximal strain to Potamoge ton pecinatus L. but breaking force was not so high. In wet conditions, the breaking strain had higher val ues than in dry tests, but the breaking force had the opposite ratio. Table 1 compares results obtained from tension tests in dry and wet conditions of two gen era of Potamogeton. Elodea canadeis L. had similar values of breaking strain and force to values acquired in tests with Ceratophyllum demersum L. but because only 20 stem pieces were tested, it was hard to indi cate true differences between tests conducted in dry or wet conditions. Callitriche palustris L. tests showed no valuable data, because most of the specimens were tore apart at the very beginning of the test.

3.2 Bending tests

Myriophillum spicatum L. showed very different results from tests conducted in dry and wet conditions. All specimens had similar diameter, slightly above 2 mm. In the dry conditions twice as many specimens were tested as in wet conditions. The maximal force used to bend a plant stem was twice as high in the case of wet conditions as it was in dry conditions. This behaviour is connected with three things that occurred during wet tests. The first was that buoyancy acted on the submerged plant, the second was that submerged plants did not slip from the bars and the third was that the pressure in plant tissue did not decrease because of water evaporation, and therefore stems were stiffer and more resistant. For Potamogeton pecinatus L. only dry tests were conducted because during the second week of testing when dry tests were planned, it was found that collected specimens were too thin and frag ile for bending tests. From the first week of testing, 30 specimens were bent in dry conditions and the maximal force and the corresponding deflection were almost identical to those of Myriophillum spicatum L. in exactly the same test.We concluded that tests carried out in wet conditions should also give very different results, giving a much better perspective on forces acting on aquatic plants. During none of the bending tests was any specimen fractured. 4 DISCUSSION During preparation of specimens, the changes in plants' internal structure were observed with a microscope. Those changes that occurred were caused by water evaporation from plant tissues, making stiff stems more limp and reducing hollow spaces within aerenchyma. One hour after removal of plants from the water tank, changes in diameter and shape were visible to the naked eye. A drop of turgor in a stem affects the plant in such a way that cracks in tissue occur faster. Because of that, stem ruptures easily, which causes the values of maximal strain to be lower. Overall, the time taken during the test is crucial to prevent tissue deformation owed to drying of plant tissue. During the tests four photographs of stem cross-sections of Potamogeton pecinatus L. were taken by microscopic camera. In a short period of time (about 20 seconds) stem diameter shrank drastically. This effect is shown in Figure 5. Dry specimens are easier to handle and fix in the machine. Also, there must be appropriate scaffolding for tests conducted in wet conditions, to ensure proper fix between clamps or between bars in bending tests so the specimens cannot slip away and emerge from water. The biomechanical tests carried out in wet conditions had to consider displacement of stems in water. During the tension test, buoyancy force was simply neglected by resetting the force to a zero value, when the stem had already been fixed in both clamps. During the bending tests that procedure was also implemented but because the stem was floating freely, it could rebound from the bending bar so the buoyancy force could be added again to a measurement. Nevertheless, the buoyancy force acting on a bending head was rather small, about 0.002 N, which is ten times lower than the minimal obtained value of force from all tests. When a whole plant is tested, the buoyancy force will be considerably greater and in proportion to the volume of space in water it occupies. Before the tests began, a few ways of correctly fixing stems to machine clamps were checked. UVhardened glue, two-substrate mixed glue, epoxy glue and artificial jaw-fixing glue did not harden fast enough to prevent specimens from drying. Superglue turned out to be the quickest and only a little amount of it was needed for each specimen. However, the glue was also watery before hardening and leaked sometimes from between sandpaper. That caused the situation where glued parts of a stem were stiffer and harder than the raw stem. Because of that, a tendency to fracture in the junction point between the gluecovered and the raw part of the stem was observed. An example of a properly fractured specimen is shown in Figure 6. Comparison of maximal force between

Figure 5. Diameter changes of stem cross-section of Potamogeton pecinatus L. within 20 seconds, where solid bottom line

is always equal 1.21 mm.

Figure 6. Example of fractured specimen of Potamogeton pecinatus L. in tension test.

specimens where such a breaking point occurred and those specimens where a fracture occurred somewhere near the middle of a stem showed no significant differ ence. Nevertheless it is recommended to use a minimal amount of glue to avoid hardening the stem surface and changing its stiffness.

Another issue that should be taken into consider ation is the question of how to test a whole beam of stems in the case of very dense branched plants like Callistriche Palustris L. Its stems are fragile but tan gled and in flowing water they act as one entity. The cumulative resistance to flow allows this species to thrive in rivers. Therefore, tests for that species should be carried out for many stems and stress values calcu lated as a force acting on the sum of cross-sectional areas of individual stems.

Individual characteristics of aquatic plants species are basis to understand patterns of how plants grow in rivers and how they alter the flow characteristics. More stiff stems result in greater pressure gradients in water around submerged plants. In addition, the stiff ness and motion of plants are crucial to obtain value of drag force exerted on plants in flowing water. Finally,

results received from tension and bending tests and field measurements of turbulent flow around plants

Jordan, B.M. & Dumais, J. 2010. Biomechanics of plant cell growth. Encyclopedia of Life Sciences. Chichester: John Wiley & Sons.

Kałuz˙a, T. & Tymin´ski, T. 2010. Analysis of the flow resistance in zones with flexible vegetation (in Polish). Infrastruktura i Ekologia Terenów Wiejskich 8(1): 79–91.

Koehl, M.A.R., Silk, W.K., Liang, H. & Mahadevan, L. 2008. How kelp produce blade shapes suited to different flow regimes: a new wrinkle. Integrative and Comparative Biology 48: 834–851.

Madsen, J.D., Chambers, P.A., James, W.F., Koch, E.W. & Westlake, D.F. 2001. The interaction between water move ment, sediment dynamics and submersed macrophytes. Hydrobiologia 444: 71–84.

Miler, O., Albayrak, I., Nikora, V., Crane, T. & O'Hare, M. 2010. Biomechanics of aquatic plants and its role in flow-vegetation interactions. Proc. River Flow 2010 Braunschweig 1: 253–260.

Miler, O., Nikora,V.,Albayrak, I. & O'Hare, M. 2012. Biome chanical properties of aquatic plants and their effects on plant-flow interactions in streams and rivers. Aquatic Sciences 74(1): 31–44.

Miler, O., Nikora,V.,Albayrak, I. & O'Hare, M. 2014. Biome chanical properties and morphological characteristics of

lake and river plants: implications for adaptations to flow

conditions. Aquatic Sciences 76(4): 465–481.

Moulia, B. 2013. Plant biomechanics and mechanobi

ology are convergent paths to flourishing interdisciplinary research. Journal of Experimental Botany 64(15): 4617–4633. Nepf, H.M., 2012. Hydrodynamics of vegetated channels. Journal of Hydraulic Research 53: 262–279 Niklas, K.J. 1992. Plant Biomechanics. An Engineering approach to plant form and function. Chicago: University of Chicago Press. Nikora, V. 2010. Hydrodynamics of aquatic ecosystems: an interface between ecology, biomechanics and environmental fluid mechanics. River Research and Applications 26: 367–384. Puijalon, S., Bourna, T., Douady, C., Van Groenendael, J., Anten, N., Martel, E. & Bornette, G. 2011. Plant resistance to mechanical stress: evidence of an avoidance-tolerance trade-off. New Phytologist 191: 1141–1149. Schutten, J., Dainty, J. & Davy, A.J. 2005. Root anchorage and its significance for submerged plants in shallow lakes. Journal of Ecology 93(3): 556–571. Siniscalchi, F. & Nikora, V. 2013. Dynamic reconfiguration of aquatic plants and its interrelations with upstream turbulence and drag forces. Journal of Hydraulic Research 51(1): 46–55. Sukhodolov, A.N. 2015. Field-based research in fluvial hydraulics: potential, paradigms and challenges. Journal of Hydraulic Research 53(1): 1–19. Wojtaszek, P. 2011. Mechanical integration of plant cells and plants. Series: Signaling and Communication in Plants. Berlin, Heidelberg: Springer-Verlag. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

LS-PIV procedure applied to a plunging water jet issuing from an overflow nappe

Y. Bercovitz, F. Lebert & M. Jodeau

EDF R&D National Hydraulic and Environment Laboratory (LNHE), Chatou, France

C. Buvat & D. Violeau

EDF R&D National Hydraulic and Environment Laboratory

(LNHE), Chatou, France

Saint-Venant Laboratory for Hydraulics, Chatou, France

L. Pelaprat

Ecole Nationale des Travaux Publics d'Etat, Villeurbane, France

A. Hajczak

Ecole Nationale Supérieure de Mécanique et d'Aérotechnique, Poitiers, France

ABSTRACT: The aim of this work is to understand the behaviour of a sheet of water falling freely under gravity, subject to instability and breaking up into droplets. Such behaviour significantly affects the pressure force at the toe of large dams equipped with ogee-type weirs. In this laboratory experiment, the LS-PIV (Large Scale – PIV) procedure is applied to determine the surface velocity field of a water jet issuing from a sharp crested weir. The fall is 5 meters high and the weir is 40 cm wide with a maximum discharge flow of 0.050 m^3/s.

In order to obtain high resolution measurements, the water jet was analysed using a high speed video camera (1000 fps). The initial results have been post-processed with the FUDAA-LSPIV software, co-developed by EDF and IRSTEA. The patterns of deformation of the water-air interface were recorded to determine the velocity vectors. The deformation patterns are relatively small thanks to the high frequency acquisition of the video camera. No special tracers were required as the high frequency acquisition ensures a good correlation factor is consistently obtained. This method allows for continuous monitoring of a large velocity field at a high time frequency.

The discussion covers details of the corrections that were

made in order to integrate the nappe trajectory and

presents the finding of a characteristic distance following which energy dissipation becomes significant.

1 INTRODUCTION

1.1 Context

The accumulation of statistical data and scientific advances in hydrology gives rise to re-evaluations of the extreme flood discharges that can transit hydro electric facilities. Moreover, regulations are evolv ing to ensure greater control over safety issues, an example of which could be an increase in the number and frequency of extreme events taken into account for the design of hydroelectric projects. It is therefore important to gain a better understanding and knowl edge of the physical processes necessary for the design of protective measures.

The study discussed in this paper applies notably to high (flood) water releases of the spillway type. In a general manner, we are looking to characterize the hydrodynamics of a rectangular jet of water issuing from a spillway. The water/air interactions along the jet are assumed to be sources of energy dissipation and better knowledge of their influence on the flow could reduce conservatisms in the analysis of estimated stresses at the base of the structure induced by overtopping flow. Our experimental setup notably allowed high speed films (1000 frames per second) to be taken of the rectangular, free-falling jet. The observations made from these video recordings are supplemented by image analysis with the LS-PIV (Large Scale Particle Image Velocimetry) method, using the free software, FUDAA-LSPIV (Le Coz et al. (2014)), in order to obtain velocity fields for the jet. The resulting measurements are compared against analytical solutions drawn from the works of Castillo (2006). The present article describes the setup and methods deployed in the laboratory experiments and presents the first results. 1.2 Previous research A two-phase flow approach to free falling jets introduces complex phenomena which combine turbulence

Figure 1. Reference frame.

and surface tension. This means of course that partic ular attention must be paid to scale effects but also increases the difficulty of applying a comprehensive approach to the forces exerted on the flow.

The study of jet energy dissipation in a plunge pool and its erosive power has been the subject of numer ous publications, including Moore (1943), Albertson et al. (1950), Horeni (1956), Lencastre (1961), Cola (1965), Castillo (1989, 2006, 2007), Withers (1991), Puertas (1994), Ervine et al. (1997), Puertas & Dolz (2005), Bollaert (2002), Bollaert & Schleiss (2003, 2003), Manso et al. (2005, 2008), Melo et al. (2005), Carrillo (2014), Carillo & Castillo (2014). However, only a few of these authors have sought to charac terize the hydrodynamics of a free-falling jet: Horeni (1956), Whiters (1991), Puertas (1994), Ervine & Falvey (1987), Ervine et al. (1997), Castillo (2006, 2007), Carillo & Castillo (2014).

One characteristic quantity in the dispersal of jets is the breakup length, (L_b), which corresponds to the

distance of fall after which point the jet is no longer

composed of anything but smaller or larger droplets of water. Drawing on the works of Horeni (1956), Ervine (1997) and Castillo (2006), the following expression of L_b is proposed for rectangular jets: With L_b in m, B_i the half-thickness of the jet in m, F_i the Froude number and T_u the turbulent intensity at a point in the issuing flow that corresponds to a drop of two times the head height over the weir. The discharge velocity at this point, V_i, is assumed equal to gravitational fall velocity. We assume $V_i = \sqrt{2g2H}$ with g the acceleration of gravity in m^2/s and H the height in m of the head of water at the crest. For a standard sharp crested weir, again according to Castillo (2006), H can be estimated by the following formula:

Figure 2. Global overview of the experimental setup.

With q the linear discharge in $m^3.s^{-1}.m^{-1}$ and C_d the

discharge coefficient approximately equal to 1.85 in

this case. The coordinate system is here C1 shown on

Figure 1, as are the quantities H, B_i and V_i.

2 EXPERIMENTAL SETUP

2.1 The spill tray

The experimental setup is composed of a tray, raised

to a height of 5 m above the ground and discharging

into a receiving basin. The tray is 1 m in length and

40 cm wide. The discharge is distributed over the full

span of the tray, with a height of the weir of 40 cm (See

Figure 2). Graduated scales provide a means of esti

mating the position of the jet in the three dimensions

of space.

Gates are installed immediately downstream of the spill tray to prevent lateral dispersion of the jet which

would have added another undesirable scale effect.

The maximum available discharge is 0.080 m^3/s,

equivalent to 0.200 $m^3.s^{-1}.m^{-1}$. For the study

presented here however, we have focused on

four discharge rate values: 0.0255 $m^3.s^{-1}.m^{-1}$,

0.050 $m^3.s^{-1}.m^{-1}$, 0.075 $m^3.s^{-1}.m^{-1}$ and 0.1005

$m^3.s^{-1}.m^{-1}$.The rate of discharge is controlled by way

of an electromagnetic flowmeter installed on the sup

ply pipes.The discharge arrives in the tray after passing

through a filter wrapped in a permeable geotextile and,

immediately afterwards, a diffuser grid, installed just

downstream of the filter. This arrangement ensures

the homogeneous supply of water to the spillway tray Figure 3. Snapshot at 75 l/s/m from the high-speed films. and provides the opportunity to make adjustments, if necessary, to the turbulent intensity of the flow. 2.2 Measurement devices The turbulent intensity, T_u, for which the expression is recalled below, is obtained by measuring the flow velocity over the spill tray with an ADV probe. $\bar{U}$ is the mean velocity at the measurement point in $m.s^{-1}$. u', v' and w' are the three fluctuating components of the velocity in $m.s^{-1}$. The results that are presented hereafter were obtained for a turbulent intensity of 16% measured 2 cm above the crest. It was not possible to take these measurements for q = 0.025 $m^3.s^{-1}.m^{-1}$ because the thickness of the nappe was not sufficient to install the probe. The high-speed camera used in the study is the FASTCAM SA3, which can record up to 2000 fps at a resolution of 1024 × 1024 pixels. 3 VISUAL OBSERVATIONS The most striking finding from observation of the high-speed films (see Figure 3) is that there are no

Table 1. Lb as a function of the linear flowrate. L_b (m) L_b (m)

q ($m^3 . ^{-1} .m^{-1}$)	for T_u = 16%	for T_u = 1.2%
0.0255	0.05	0.41
0.05	0.08	0.64
0.075	0.10	0.86
0.1005	0.12	1.03

Figure 4. Overflow at 0.025 $m^3 s^{-1} m^{-1}$ (a) side view (b) front view.

air bubbles in the water. The jet deforms then breaks up into filaments and droplets leading to the appear ance of cavities. The white water effect that is seen by the naked eye and is captured in Figure 2 is, in fact, the result of the integrative processes of the brain which perceives the rapid, diffused scattering of reflected light from the water surface as a single blur of white ness (for the photo, this effect is linked to the exposure time). Although this could have led to the perception that we had found similar entrainment phenomenon in the jet to those occurring in flow down a steep chute for example, it is clear that such an approach would lead to serious and potentially dangerous errors.

The second observation of major significance is that, despite the high turbulent intensity measured over the spillway, there is no clear evidence of flow break up for the three biggest studied fows : an uninterrupted flow path is always found between the bottom of the jet and the point of discharge. However, applying equation (2) under the same assumptions established by Castillo (2006) and (2014) results in the values listed in Table 1.

When $q = 0.025.m^3.s^{-1}.m^{-1}$, a lateral contraction is observed in the jet (Figure 4 (b)). Moreover, this overflow nappe falls within the scope of thin liquid sheet oscillating flows (Figure 4 (a)). The characterization of these oscillations has been the subject of numerous studies: Squire (1953), Jazayeri & Li (2000) and Tharakan et al. (2002) to name but a few. In this particular case, the break-up occurs at a fall distance of around 3 m. 4 LS-PIV MEASUREMENTS As deformation pattern movements were visible on the video (Figure 3), we chose to use the LS-PIV technique in order to obtain the velocity field in the jet. The free software FUDAA-LSPIV has been used. The results were post-processed using FUDAA-LSPIV, which is a free software co-developed by EDF and IRSTEA. This notable advantage of this tool is that it includes an orthorectification module for image processing, Hauet (2006). This process serves to transform images such that distances and hence velocities are directly measurable on a reference (or datum) plane. The latter is usually fixed at the level of the free surface. 4.1 Parameterization In order to obtain the velocity field along the jet, we recorded the flow over a drop of nearly 3.5 m with an image acquisition rate of 1000 frames per second. Using the premise that, at this rate of acquisition, the deformation processes occurring in the nappe would be sufficient to track the flow, no tracers were used. An interrogation window size of 12 pixels, with one pixel representing an area of 4 mm^2, was used to search for the target patterns. The minimum correlation parameter was set to a value of 0.6. Post-processing was systematically carried out on 1,999 pairs of images; as the time interval separating two consecutive images is 0.001 s, the processing of each video is completed in 2s real time. We thus have 1,999 velocity measurements for each point of the grid, with the latter containing 200 points. 4.2 Trajectory of the jet FUDAA-LSPIV was initially developed to obtain flow measurements in river applications, for which the frame of reference (or free surface) is close to the horizontal plane.We therefore had to add a patch to account for the trajectory of the nappe. We are going to present here two linear interpolation method to estimate the trajectory, an analytic one and an empiric one. Scimeni (1937) proposed the following equation to determine the trajectory of a falling nappe below the inflection point which is here the coordinate systeme origin (C2 Fig.1): With z the vertical coordinates in m and x the coordinates in m along the longitudinal axis. We then applied a linear interpolation, after the first meter drop, for

Table 2. Slope coefficients of the linear regressions for the

discharge trajectory measurement plots.

q ($m^3.s^{-1}.m^{-1}$)	α	β
0.0255	-11.70	3.42
0.0500	-9.68	3.37
0.0750	-7.40	2.96
0.1005	-7.26	2.94

each of the discharge rates to define the frame of

reference.

The curvature of the nappe in the first metre drop

is large and our measurements are contaminated with

errors in this area however, beyond this point, the linear

approximation seems valid. We assume after the first

meter drop the thickness of the jet is negligible for

this approximation of the trajectory. For a monophasic

approach, in the coordinate system C1 (Fig. 1) we have

Which gives a thickess lower than 0.023 m for the

studied flows.

The reference plane required for the LS-PIV pro cessing passes through the line thereby obtained and is orthogonal to the plane (x,z), which correspond to the plane (y,z) in the coordinate system C3 (Fig. 1). The velocity fields are obtained in this plane and tend to be parallel to the discharge velocity.

Unfortunately, this analytical approach did not prove entirely satisfactory for estimating the discharge trajectory, notably for $q = 0.100\ m^3\ s^{-1}\ m^{-1}$, where our measurements of the fall velocities appear to be largely underestimated. The analitycal approach need to be corrected. So we decided measure the trajectory for each discharge by filming the flow and then apply ing the FUDAA-LSPIV orthorectification technique to the resulting images. Figure 5 illustrates the differ ence between our measurement results and the Scimeni profiles. For this figure, the origine of the coordinate system is the crest of the weir. The free-surface ele vation, which becomes very low after a one-metre fall (the order of magnitude being a few mm) is not taken into account here. It should be noted that our measurements no longer fall within the scope of appli cation of the formula established by Scimeni (1937) who did not consider the two-phase nature of the flow and whose experiments were conducted in channels, largely limiting air circulation because his fall was largely smaller than our one. We deliberately chose

to present the results in Figure 5 in dimensional form

because they seem to be highly dependent on the

experimental conditions, in particularly the turbulent

intensity.

Table 2 provides the slope coefficients (α) and the

y-intercepts (β) for each of the regression lines.

For Figure 5 and Table 2 the coordinate system is

C4 (Fig. 1).

Furthermore, the jet was first filmed at a distance

of around 6 m in order to capture the entire fall but the velocities appeared largely underestimated. We thus moved the camera closer so as to film approximately 3 m of the fall. 4.3 LS-PIV results Based on the corrections discussed beforehand, we were able to obtain the velocity, at the centre of the jet, the results are presented on Figure 6. The jet discharged at $q = 0.0255\ m^3.s^{-1}.m^{-1}$ was not sufficiently deformed to obtain effective measurements without tracers, consequently they are not presented here. Here, the z coordinate is expressed in the coordinate system C1 (Fig. 1). These results indicate a close fit between the measured velocities and the velocity profile of gravity drop flows up to a point that corresponds to a fall of about 3 m, beyond which the measured velocities drop off. Visual observations reveal that the jet broadens at this distance, which explains this sudden, rapid deceleration. Thus, without having observed, in the strictest sense of the word, the break-up we demonstrate empirically that there is a characteristic length beyond which the velocity diminishes. In the laboratory experiments, this characteristic length is much greater than the breakup length obtained using the Castillo (2006) equation. A reasonable explanation has not as yet been found for these differences. Before dropping away, the measurements did not always correspond perfectly to the gravity velocity profile although this may be explained by the slight imprecision of our trajectory measurements and by their linear approximations. Strictly speaking, the nappe curvature should be taken into account in the orthorectification process. 5 CONCLUSION Better knowledge of the dynamics of jets in falling nappes is becoming an industrial necessity but the processes called upon by this area of physics are numerous and their interactions complex. The experimental set up presented herein analyses four discharges for a total drop height of 5 m. We have proposed a first analysis based on visual observations of high-speed video recordings. We found no evidence of air bubbles penetrating the jet, however the latter progressively decomposes, as a result of which cavities of various sizes appear. We did not manage to clearly identify the disintegration of the jet (breakup) as although the geometry of the falling nappe becomes very sophisticated, we were always able to find a continuous flow path joining the point of discharge to the bottom of the chute. We then attempted to measure the discharge trajectory. It seems that the two-phase nature of the flow has a significant influence on this trajectory. Beyond a certain drop height, the approach taken by Scimeni

Figure 5. Dimensional comparison of the Scimeni (1937) profiles and the experimental measurements (a)

$q = 0.025$ $m^3.s^{-1}.m^{-1}$ (b) $q = 0.050$ $m^3.s^{-1}.m^{-1}$ (c) $q = 0.075$ $m^3.s^{-1}.m^{-1}$ (d) $q = 0.100$ $m^3.s^{-1}.m^{-1}$.

(1937) is no longer adequate and tends to overestimate

the outward projection of the jet.

To obtain the profile of the fall velocity along the

jet we propose an approach that uses the LS-PIV tech

nique. This method allows us to address a large part of the drop with a single shot. Initial recommendations for the measurement parameter settings have been put forward and we have drawn the attention of the reader to errors that should be avoided should intend to implement a similar test bench.

Figure 6. LS-PIV results (a) $q = 0.050$ $m^3.s^{-1}.m^{-1}$

(b) $q = 0.075$ $m^3.s^{-1}.m^{-1}$ (c) $q = 0.1005$ $m^3.s^{-1}.m^{-1}$.

For the discharges considered in the study, the LS

PIV measurements follow the profile of free fall (or

gravity) velocity over the first three metres but, beyond that, the velocity distribution becomes highly vari able and decreases sharply. While not corresponding exactly to the physical definition of break-up, we reveal thereby a characteristic length of the jet beyond which energy dissipation would become significant.

The LS-PIV measurements could and should be improved. To achieve this, it is necessary to bet ter understand and control the jet trajectory, taking account notably of its curvature. Moreover the time averaged velocities were obtained over 2 s intervals, which is probable insufficient.The use of tracers could enable better capture of the fall velocities and would

Le Coz J., Jodeau M., Hauet A., Marchand B. & Le Boursi caud R. 2014. Image-based velocity and discharge mea surements in field and laboratory river engineering studies using the free Fudaa-LSPIV software. Int. Conf. River Flow 2014. Lausane, Switzerland 1961–1967.

Lencastre A. 1961. Descarregadores de lãmina livre: Bases para o seu estudo e dimensionamento. Memoria Nº 174. Laboratorio Nacional de Engenharia Civil, Lisbon.

Manso P.A., Bollaert E.F.R. &Schleiss A.J. 2005. Dynamic pressures generated by plunging jets in confined pools under extreme flood discharges. Proc. XXXI IAHR Congress, Seoul, 1, 2848–2860.

Manso P.A., Bollaert E.F.R. & Schleiss A.J. 2008. Evaluation of high-velocity plunging jet-issuing characteristics as a basis for plunge pool analysis. J. Hydraulic Res. 46(2), 147–157.

Melo J.F., Pinheiro A.N. & Ramos C.M. 2005. Forces on Plunge Pool Slabs: Influence of Joints Location and Width. J. Hydraulic Eng. 132(1), 49–60.

Moore W.L. 1943. Energy loss at the base of a free over all. Transactions, American Society of Civil Engineering, 108(1) : 1343–1360.

Puertas J.. 1994. Criterios hidráulicos para el diseño de cuencos de disipación de energía en presas bóveda con vertido libre por coronación. PhD Thesis. Departamento de Ingeniería Hidráulica, Marítima y Ambiental, Universidad Politécnica de Cataluña, Spain.

Puertas J. & Dolz J. 2005. Plunge pool pressures due to falling rectangular jet. J. Hydraulic Eng. 131(5), 404–407.

Scimeni E. 1937. Il profile delle dighe sfioranti. L'energia Elettrica. 14, 937–940.

Squire H.B. 1954. Investigation on the instability of a moving liquid film. British journal of Applied Physics. 6(4), 167– 169.

Tharakan T.J., Ramamurthi K., Balakrishnan M. 2002. Nonlinear breakup of thin liquid sheets. Acta Mechanica. 156, 29–46.

Withers W. 1991. Pressure fluctuation in plunge pool of an impinging jet spillway. PhD Thesis. Department of Civil Engineering, University of Glasgow, United Kingdom.

Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

PIV-PLIF characterization of density currents

B. Pérez-Díaz

Environmental Hydraulics Institute "IH Cantabria", Universidad de Cantabria, Spain

P. Palomar

Environmental Hydraulics Institute "IH Cantabria", Universidad de Cantabria, Spain

Ministry of Agriculture, Food and Environment, Spain

S. Castanedo

Environmental Hydraulics Institute “IH Cantabria”, Universidad de Cantabria, Spain

Departamento de Ciencias y Técnicas del Agua y del Medio Ambiente, Universidad de Cantabria, Spain

ABSTRACT: Saline density currents are horizontal flows driven by the density difference between the environmental fluid and the density current. In recent years, as the problems related to environmental conservation and coastal development have become more serious and complicated, it has become important to understand the behaviour of these flows. This work carries out an experimental characterization of saline density currents through advanced non-intrusive laser optical techniques PIV (Particle Image Velocimetry) and PLIF (Planar Laser Induced Fluorescence). By means of synchronized PIV-PLIF techniques, high-quality accurate instantaneous measurements of velocity and concentration are obtained. The aim of these experiments is to study the quasi-steady flow properties of the current body generated by a constant flux release. Seeking elucidation about the most influential variables in the behaviour of density currents, different experimental set-up varying the initial conditions (flow rate, thickness, slope, excess density) were carried out in a 3 × 3 × 1 m tank. Through PIV-PLIF result analysis, important conclusions about the influence of these variables on the mixing at the interface between fluids have been obtained. To carry out a quantitative comparison between currents, the stable mixing rate along each current is evaluated, which is commonly known as Entrainment (E) in scientific literature. As an example, keeping constant the rest of variables, steeper slopes and higher flow rates favour mixing, i.e.

dilution, reaching stable mixing rates (E) values two times higher than values obtained in the corresponding base case ($E \sim 2 \cdot 10^{-2}$). In addition, a high resolution and quality experimental database has been generated, which will allow to calibrate/validate hydrodynamic modelling tools.

1 INTRODUCTION

There are numerous natural and industrial occurrences of density current motions (Simpson, 1982, 1997; Huppert, 2006) caused by different kind of discharges. Several classifications of these horizontal flows can be made in broad terms regarding its general behaviour and the fluid nature. Taking into account the sign of the density difference between the ambient fluid and the current, hence the different ways of interac tion between the two fluids through its boundaries, density currents can be classified in “bottom den sity”, “free-surface density” and “neutral density” currents. Bottom density currents have been widely examined (Simpson, 1982; Chowdhury & Testik, 2014), and have been classified according to their fluid nature in turbidity (particle-leaden) and saline current.

This study focuses on the density currents gen erated by hypersaline discharges from desalination plants (Palomar & Losada, 2010), commonly preceded by a more turbulent phase described and studied in Palomar et al. (2012a, b).The specific interest of studying these currents steams from the fact that these slow flows creep along large areas of the seabed, which could damage marine ecosystems even far from the discharge point. The complexity of such flows, the lack of experimental and field data and the absence of reliable modelling tools have motivated the performance of several experimental tests, within the framework of SALTYCOR I + D + i National Plan of Spain Ministry. These experiments have been carried out inThe Environmental Hydraulics Institute of Cantabria's facilities using PIV and PLIF. These techniques provide high-quality accurate instantaneous measurements of velocity and concentration (Martin & García, 2008; Liao & Cowen, 2010). The present work shows the main results of the experimental database analysis, which main objective is to elucidate the behaviour of three-dimensional constant-flux saline density currents. Experimental set up and procedures are described in Section 2, results are presented and analysed in Section 3, and their final conclusions in Section 4.

Figure 1. Laboratory photographs of the saline density cur

rent and the discharge device. The measurements PIV-PLIF

were taken along the coordinate plane XZ.

2 EXPERIMENTAL SET-UP

Experiments were conducted in a 3 × 3 × 1 m 3 test

tank filled with freshwater to simulate the receiving

body, a 1000 lt plastic effluent storage tank and a

100 lt steel constant head tank, containing both the

effluent (mixing of fresh water, salt, dye tracer and

small particles). The storage and the constant head

tanks were connected to each other by a pump for re

circulating the flow and ensuring a steady flow. The

effluent was discharged by a gravity-driven force from

the constant head tank into the test tank, measuring and controlling the flow rate by an electromagnetic flow-meter and a precision valve, respectively. The test tank was made of steel with two lateral glass win dows: one is used to illuminate the laser sheet and the other one allows taking images. The walls, bot tom and aluminium discharge tube were painted in black to avoid laser reflections. A plastic plate, to sim ulate the seabed, was installed inside the test tank, 30cm over the bottom and 20 cm from the side walls. A trap was designed between the plastic plate and the test tank bottom, to allow the brine to be stored during the experiment, thus preventing tank contami nation and allowing the study of non-confined gravity currents (i.e. three-dimensional). Figure 1 shows two photographs of the discharge device, a transparent methacrylate thank with a height-adjustable slot at the bottom. Figure 2 shows the scheme of the experimental set-up.

Two Imager ProX 4M (CCD) cameras with 2048×2048 pixel 2 resolution were used to record the PIV and PLIF images. In order to capture the maximum covering area, around 1400 mm in the lon gitudinal direction regarding the flow, both cameras were located adjacently with an overlapping zone. Figure 2. Schematic figure of the experimental set-up. Source: Palomar (2016). Thus, each PIV and PLIF image covered approximately 700 mm. In these conditions, 200 pixels are available to cover the gravity current in concentration fields, whereas 12 velocity vectors characterised velocities along the density current. This resolution together with the appropriate selection of PIV and PLIF test parameters enabled the accurate measuring of concentrations and velocities. The selection of PIV and PLIF parameters were based on previous works as Keane & Adrian (1990), Ferrier et al. (1993), Willert (1996), Crimaldi (2008) and Palomar (2016). Table 1 summarises experimental conditions and initial flow parameters (dimensionless parameters, length scales and initial fluxes) of the cases analysed in the present work. To establish these case studies some preliminary experiments were carried out. Their goal was to find the initial flow characteristics that allowed to undertake good PIV-PLIF measurements (e.g. having enough thickness, avoiding contamination) at the same time as the flow evolves in a density current with a plume-like behaviour. CASE1 is the base case on which all other cases were modified in one of the fundamental variables (thickness h_0, flow rate Q_0, slope α or density difference ($\rho_a - \rho_o$).

3 RESULTS

This section presents flow measurements from PIVPLIF experiments of Table 1 and their analysis. To characterize the hydrodynamic and mixing processes, the flow fields of main variables and the longitudinal and cross profiles along the saline density current are shown in the following subsections.

3.1 Flow fields

Instantaneous flow fields of velocity and concentration are the result of proper interpretation and coupling of each pair of adjacent images. Figure 3 shows an example of concentration flow fields of C1. Thanks to the high PLIF resolution, these instantaneous images allow to observe flow details as Kelvin-Helmholtz

Table 1. Configurations tested to characterize saline density currents

Properties ↓ / Cases →	C1	C2	C3	C4	C5	C6
Slot dim. (m) b_o	0.100	0.100	0.100	0.100	0.100	0.100
h_o	0.025	0.015	0.025	0.025	0.025	0.025
Water depth, H_a (m)	0.460	0.460	0.460	0.420	0.360	0.460
Slope angle, α (%)	1.00	1.00	1.00	2.50	4.50	1.00
Density difference, ($\rho_a - \rho_o$) (kg/m^3)	3.145	3.100	3.130	3.070	3.140	11.08
Discharge flow rate, Q_o (l/min)	14.60	15.10	19.20	14.98	14.09	14.89
Discharge velocity, U_o (m/s)	0.095	0.164	0.125	0.098	0.098	0.097

Figure 3. Snapshot normalised concentration images of C1 from Camera 1

instabilities along the interface between the current and the surrounding water.

The aim of this study is the characterization of

saline density currents once they have reached their

stationary analysis in the covering area (1400 mm),

i.e. when their flow properties are almost constant

since they are generated by a constant flux discharge.

Hence, the interest of this study is the stationary den

sity current body instead of the current front. Mean

(time-averaged) flow fields were obtained from instan

taneous images by applying the following expressions:

being: U_x and U_z , horizontal and vertical components

of mean velocity; u_{xi} and u_{zi} , instantaneous values of

horizontal and vertical velocities; C, mean concentra

tion; c_i , instantaneous values of concentration; N , the

number of images; S, the mean dilution; C_0 , initial con

centration; and C_a , surrounding fluid concentration. Figure 4. Normalized flow fileds of C1 from Camera 1: a) horizontal mean velocity; b) mean concentration. As an example, Figure 4 shows the mean concentration (C) and the horizontal component of the mean velocity (U_x) results of C1. According to mean values of velocity and concentration (dilution) calculated for all cases of study, some general conclusions can be drawn. Vertical component of the mean flow can be considered negligible compared to horizontal component. A steep decrement of the horizontal mean velocity along the underflow can be observed, this is due to the bottom friction, the surrounding stagnant fluid and the lateral spreading of the plume. Similarly to velocity, mean concentration rapidly decrease along the current, being very low downstream. This reduction is due to entrainment of the surrounding fluid into the current layer. 3.2 Longitudinal and cross profiles By taking advantage of PIV-PLIF techniques over classical spatial dispersed measurements, this study characterises the density current behaviour by means of continuous spatial evaluation. Thus, this section presents longitudinal profiles (see Figure 5) of normalized horizontal maximum velocity U_x/U_0 (being U_0 the initial discharge velocity, see Table 1) and minimum dilution S_{min} (minimum values of equation (4) results). Therefore, these longitudinal profiles

Figure 5. Longitudinal profiles of normalized horizontal maximum velocity and minimum dilution for cases: C1, C3, C5 and C6.

represent the lines that join the points of maximum mean velocity and minimum dilution of cross sections. Figure 5 reveals that the normalized maximum veloc ity and the concentration (as the inverse of dilution) decreases rapidly along the first 400-600 mm until reaching a nearly constant rate values. It can be set that three-dimensional saline density currents stud ied in this work show a "Normal Status" at certain downstream distance from the discharge slot.

The dilution rate is commonly known as Entrain ment (E) in scientific literature. Many researches are focused on the entrainment assumption, first published by Morton et al. (1956), which states that the entrain ment velocity across the interface (w_e) is proportional to the characteristic velocity of the flow (U):

These researches deduced several entrainment parameterizations from laboratory studies for differ ent kind of density currents (Ellison & Turner 1959, Parker et al. 1987), all of them in the form $E = E(R_i)$, where R_i is the dimensionless Richardson number:

being: h, u and ρ the thickness, mean velocity and

density of the density current, respectively; ρ_a, the

density of the surrounding fluid, and α the slope angle

of the bottom. R_i expresses the ratio of the potential

energy term (buoyancy) to the kinetic energy term

(shear stress). High values of R_i mean low rates of

dilution, being negligible for $Ri > 1$. Some parameter

izations are gathered in Chowdhury & Testik (2014).

In order to estimate the entrainment coefficient from

the flow measurements once the "Normal Status" is

established (E_N), a method based on the equations of

conservation of volume (Dallimore et al. 2001) has

been used. The experiments carried out reveal similar

Figure 6. Similarity cross profiles of C1: a) mean concentration; b) mean horizontal velocity. Symbols represent measurements and the line is the best fitting curve.

E_N values to those at the same regime (small R_i numbers, $R_i \sim [0.2 - 0.3]$) published by Ellison & Turner (1959); Fukuoka & Fukushima (1980); and Parker et al. (1986). By comparing the E_N values for cases of study, it can be concluded that steeper slopes (C4 and C5) and initial higher momentum fluxes (C2 and C3) favour the dilution of saline density current, reaching E_N values close to $5 \cdot 10^{-2}$ against base case (C1) and C6 values close to $2 \cdot 10^{-2}$. Regarding cross profiles. as previous studies have demonstrated (Gray et al. 2006, Gerber et al. 2011), the velocity and concentration cross profiles present self-similarity, i.e. the dependence on the downtream distance of the vertical distributions are reduced by the self-similarity hypothesis. According to Figure 6, both the normalized mean horizontal velocity and mean concentration cross profiles of C1 collapse well into a single profile.The latter can be extrapolated to all cases of study, therefore, common empirical shape parameters given by these collapse curves can be used to define vertical distributions.

4 CONCLUSIONS

An experimental study to

characterize the behaviour of three-dimensional saline density currents in calm receiving water has been presented using PIV and PLIF. The work has been focused on quasi-steady flow properties of the current body generated by a constant flux release. A broad range of different initial conditions (flow rate, thickness, slope and excess density) has been tested, motivated by its occurrence in nature, for example, in the far field of actual brine marine outfalls. The analysis of the flow fields reveals the horizontal component U_x of the mean velocity (various order of magnitude higher than the vertical counterpart U_z), decreases sharply along the three-dimensional density current due to the influence of friction with the surrounding fluid at rest, bottom friction and lateral spreading. Analogous results are shown about con centration fields. Mean concentration rapidly decrease along the current. Therefore, turbulent mass trans port is significant close to the discharge slot, but rapidly decays along the saline current. The latter was confirmed by the analysis of longitudinal profiles of maximum values of horizontal mean velocity and mean concentration.These longitudinal profiles reveal a "Normal Status" beyond a certain downstream dis tance from the discharge slot, at which the horizontal component of the mean velocity and the dilution rate reach a cuasi-constant value.

The aforementioned dilution rate is directly related to the entrainment of surrounding fluid into the den sity current along their interface (E). To compare different saline density currents evaluated, the stable entrainment coefficient reaches by each experiment was calculated, revealing that steeper slopes and ini tial higher momentum fluxes favour the dilution. In addition, the analysis of vertical structure reveals that these kinds of flows present self similarity properties. Throughout these laboratory experiments, a high resolution and quality experimental database has been generated, which will allow calibrate/validate complex numerical approximation as hydrodynamic modelling tools.

Chowdhury, M. R., & F. Y. Testik, 2014, A review of gravity currents formed by submerged single-port discharges in inland and coastal waters, Environmental Fluid Mechan ics, 14 (2), 265–293, doi:10.1007/s10652-014-9334-7.

Crimaldi, J. P., 2008, Planar laser induced fluorescence in aqueous flows, Experiments in Fluids, 44 (6), 851–863, doi:10.1007/s00348-008-0496-2.

Dallimore, C. J., J. Imberger & T. Ishikawa, 2001, Entrain ment andTurbulence in Saline Underow in Lake Ogawara, Journal of Hydraulic Engineering, 127 (11), 937–948, doi:10.1061/(ASCE)0733-9429(2001)127:11(937).

Ellison, T. H., & J. S. Turner, 1959, Turbulent entrainment in strati_ed ows, Journal of Fluid Mechanics, 6 (03), 423, doi:10.1017/S0022112059000738.

Ferrier, A., D. Funk, & P. Roberts, 1993, Application of optical techniques to the study of plumes in strati_ed uids, Dynamics of Atmospheres and Oceans, 20 (1–2), 155-183, doi:10.1016/0377-0265(93)90052-9.

Fukuoka, S. & Y., Fukushima, 1980, Reservoir, On Dynamic

Behavior of the Head of the Gravity Current in a Strati_ed,

in Proc. Second Int. Symp. Strati_ed Flow, IAHR, 1, pp.

224-227.

Gerber, G., G. Diedericks, and G. R. Basson (2011), Particle

Image Velocimetry Measurements and Numerical Mod

eling of a Saline Density Current, Journal of Hydraulic Engineering, 137 (3), 333-342, doi:10.1061/(ASCE) HY.1943-7900.0000304. Gray, T. E., J. Alexander, & M. R. Leeder 2006, Longitudinal ow evolution and turbulence structure of dynamically similar, sustained, saline density and turbidity currents, Journal of Geophysical Research, 111 (C8), C08,015, doi:10.1029/2005JC003089. Huppert, H. E. 2006, Gravity currents: a personal perspective, Journal of Fluid Mechanics, 554 (-1), 299, doi:10.1017/S002211200600930X. Keane, R. D., & R. J. Adrian, 1990, Optimization of particle image velocimeters. I. Double pulsed systems, Measurement Science and Technology, 1 (11), 1202-1215, doi:10.1088/0957-0233/1/11/013. Liao, Q., & E. Cowen 2010, Relative dispersion of a scalar plume in a turbulent boundary layer, Journal of Fluid Mechanics, 661, 412-445, doi: 10.1017/S00221120100 03058. Martin, J. E., & M. H. García 2008, Combined PIV/PLIF measurements of a steady density current front, Experiments in Fluids, 46 (2), 265-276, doi:10.1007/s00348-0080556-7. Morton, B. R., G.Taylor & J. S.Turner. 1956,Turbulent Gravitational Convection from Maintained and Instantaneous Sources, Proceedings of the Royal Society A: Mathematical, Physical and Engineering Sciences, (1196), 1-23, doi:10.1098/rspa.1956.0011. Palomar, P., & I. J. Losada, 2010, Desalination in Spain: Recent developments and recommendations, Desalination, 255 (1-3), 97-106, doi:10.1016/j.desal.2010.01.008. Palomar, P., J. L. Lara, I. J. Losada, M. Rodrigo & A. Alvarez, 2012a, Near field brine discharge modelling part 1:Analysis of commercial tools, Desalination, 290, 14-27, doi:10.1016/j.desal.2011.11.037. Palomar, P., J. L. Lara & I. J. Losada, 2012b, Near field brine discharge modeling part 2: Validation of comercial tools, Desalination, 290, 28-42, doi: 10.1016/j.desal.2011.10.021. Palomar, P, J. L.

Lara & I. J. Losada, 2016, PIV-PLIF experimental study of the spreading layer arisen from brine jet discharges, In prep. Parker, G., Y. Fukushima & H. M. Pantin, 1986, Selfaccelerating turbidity currents, Journal of Fluid Mechanics, 171 (-1), 145, doi:10.1017/S0022112086001404. Parker, G., M. Garcia, Y. Fukushima, & W. Yu, 1987, Experiments on turbidity currents over an erodible bed, Journal of Hydraulic Research, 25 (1), 123–147, doi: 705 10.1080/00221688709499292. Simpson, J. E. 1982, Gravity Currents in the Laboratory, Atmosphere, and Ocean,Annual Review of Fluid Mechanics, 14 (1), 213–234, doi:10.1146/annurev..14.010182. 001241. Simpson, J. E. 1997, Gravity currents: the environment an the laboratory, 258, Cambridge Univ. Press, doi: 10.2277/0521561094. Willert, C. 1996, The fully digital evaluation of photographic PIV recordings, Applied Scientific Research, 56 (2–3), 79–102, doi:10.1007/BF02249375. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Physical model tests for the construction stages of large breakwaters.

Case studies to the ports of Barcelona and Cruña (Spain)

R.M. Gutierrez, J. Lozano & J.M. Valdés

Centro de Estudios y Experimentación de Obras Públicas (CEDEX). Spain

ABSTRACT: Physical models are employed usually for testing and optimization of final breakwaters designs.

However, its application to the construction stage is less common. Nevertheless, its use facilitates the planning of

construction, as well as improving the safety of the works. Given this situation, this paper provides an overview of

the possibilities offered by physical experimentation for breakwater construction, by two case studies conducted

at CEDEX: Barcelona enlargement and Coruña Outer Port.

1 INTRODUCTION

Physical model tests are extensively employed, for

testing and optimization of final breakwaters designs.

However, its application to construction stages analy sis of these structures is less common and, therefore, less well known, but their use facilitates the design of such stages and the construction safety in general and especially that of the people involved in it.

Given this situation, the present paper provides an overview of several possibilities that offers phys ical experimentation on breakwater construction and presents two examples conducted at the Center of Ports and Coastal Studies of CEDEX: Barcelona port enlargement breakwater (Section 2) and Coruña Outer Port breakwater, which are representative of construc tion stage tests of the two most common breakwaters types: vertical and rubble mound.

2 APPLYING PHYSICAL MODELS TO BREAKWATER CONSTRUCTION

Some aspects that can be studied in the construc tion stage tests are: construction processes, temporary protection for construction progress (trunk, alignment changes and head) and maritime climate limit condi tions -waves and tidesfor works safety and equipment involved in the breakwater zone, so that from these limits work can be stopped, the area evacuated and equipment protected or removed.

These tests can be performed, depending on the aspects to study and the wave incidence (perpendic

ular or oblique), by two-dimensional (2D) or three

dimensional (3D) models. Scales used range between

1:10 and 1:50, with the largest possible being used in

order to minimize scale effects. Figure 1 shows three

examples of these tests. Figure 1. (1) Bilbao. Punta Lucero head (3D-1/30), (2) Bilbao. Punta Sollana trunk (2D-1/16) and (3) Coruña outer port trunk (2D-1/28.5 and 3D-1/30). 3 BARCELONA ENLARGEMENT. SOUTH BREAKWATER 3.1 Preliminary Figure 2 shows the layout of Barcelona port enlargement with East (E) and South breakwaters (S). In the case of the vertical South breakwater stretch (Section 2), the studies carried out by CEDEX involved 2D physical model tests to check the final section stability, and some temporary construction stages (CEDEX, 2003). This paper presents the 2D physical model tests conducted for those construction stages, which are relative only to the caisson (not including the crown parapet) with and without the cover slab, with the scope to determine the ground pressures and for measuring waves forces inside open caisson cells.

Figure 2. Barcelona Port Enlargement. Layout.

Figure 3. Barcelona South breakwater. Type section.

Figure 4. Barcelona South breakwater. Construction stages.

Section.

3.2 South breakwater typical section

The South breakwater Section 2 is a vertical structure,

1,700 m length and 32 m maximum height, founded

on ground of low bearing capacity; preliminary con

solidation was necessary by building the breakwater

in several temporary stages. The caisson is supported

by rockfill at –20 m and crown by a parapet at +12

m. Figure 3 shows the section tested.

3.3 Breakwater construction stages

Two construction stages were tested: the caisson with and without slab cover (figure 4). In the case of caisson without slab, the cell behavior was tested considering both emptied and filled with sand and gravel.

3.4 Test waves

Tests were performed with Tp = 11 and 13 s and with Hs increasing, in 0,5 m steps, from 3 to 6 m using a Jonswap spectrum (peak parameter γ= 3.3) and con sidering a mean tide level (0.0 m).The duration of each condition was set so that the number of waves gener ated was sufficient to ensure section equilibrium state (>500), with special importance for sand and gravel

cell fills. Figure 5. Large Scale Wind and Wave Flume. Overview without and with the wind tunnel and longitudinal flume profile. 3.5 Test flume The tests were performed in the CEDEX Large Scale Wind and Wave Flume (Figure 5). The main characteristics of these facilities are: 90 m long, 3.6 m wide and depth from 6÷4.5 m; wave generation rotating paddle (dry inner surface, 22.50 ∘ max. rotation, 300 kw); regular/irregular wave generation, (max. height: 1.60 m) and active absorption reflected wave system and its wind tunnel enable work to be carried out with large models, whilst minimizing scale effects occurring in tests with smaller dimensions. 3.6 Test characteristics 3.6.1 Similarity law, model scale and scale effects As is general practice, Froude's similarity was used to a geometric scale of 1:10. This was chosen considering: wave generation capacity to reproduce waves proposed; flume wave gauges operating range; model size and scale effects. The scale effects due to the elasticity forces (Fe), surface stress ($F\sigma$) and viscosity ($F\mu$) can be disregarded. In the case of Fe, this is because water can be considered as virtually incompressible in this kind of test. In the case of $F\sigma$, this is because when the periods are greater than 0.5 s, as is habitual in maritime physical models, the wave movements are governed by gravitational action not by surface stress forces, and, finally for Fμ, it is because the Reynolds model number $(Re)_m$ * is> 3×10^4 for H_D > 0.09 m, so the model flow is turbulent as in the prototype. *$(Re)_m = \lambda^{-3/2} (Re)_p$; $(Re)_p = \sqrt{g \cdot H_D} \cdot l_e / v$; H_D : H_s starting damage; $l_e = (P/\gamma)^{1/3}$ (P: block weight; γ: block density). 3.6.2 Test waves calibration Before the tests, waves were calibrated to create paddle movements that would make it possible to reproduce wave characteristics adapted to the defined JONSWAP spectrum. With a view to this, a record was generated for each of the waves and water levels whose duration was such that the number of waves was sufficient for statistical analysis.

Figure 6. Barcelona South breakwater. Section 2. Construction stages. Instrumentation.

3.6.3 Model instrumentation

For determining wave forces on the caisson: front face, base, covering slab and open-cell walls, pressure sensors were arranged as shown in Figure 6.

3.7 Test methodology

3.7.1 Wave generation and measurement

Waves generated were measured to make sure that they were in accordance with test waves proposed. Measurements were analyzed with Mansard-Funke method to calculate Hs incident and reflected.Also the GEDAP application (NRC, Canada) was used.

3.7.2 Forces measurement

Force determination on the caisson was performed by integration of measured values in the pressure sensors. Initially a 50 Hz frequency was used and afterwards a 200 Hz for higher forces.

3.8 Test results

Force time series were obtained on: the caisson

front face, base, covering slab, and open cells walls.

Figure 7 shows the pressure diagram for Tp = 11 s

and Hs = 8.0 m and maximum evolution impact for

Tp = 13 s Hs = 5 m, including the contribution inside

cells forces.

The wave force contribution on open cells (without

slab) to the overall was 24% for Hs = 4.5 m and 38%

for Hs = 5 m. Foundation pressures were similar with

and without slab for unfilled cells and higher with cells

filled and no slab.

Erosion in filling the open cells (sand and gravel)

is shown in Figure 8.

4 CORUÑA OUTER PORT BREAKWATER

4.1 Preliminary

Figure 9 shows the site of the Coruña outer port and

an overview of its large breakwater.

In the case of the breakwater, the studies conducted

by CEDEX involved many 2D and 3D physical model

tests (CEDEX, 2008 and 2010) to check the behav

ior of the section type stability, its singular sections
Figure 7. Barcelona South breakwater. Section 2. Pressures distribution (Tp = 11 s; Hs = 6.0 m) and maximum horizontal forces evolution, including open cells contribution (Tp = 13 s; Hs = 5.0 m). Figure 8. Barcelona South breakwater. Section 2. Sand and gravel erosion in open cells. (starting stretch, trunk, direction change and head), the overtopping and the various construction stages. This paper presents

the 2D physical model tests conducted at various construction stages in order to: (A) define temporary construction protections in the work platform (CEDEX, 2010) and (B) determine the waves at which works must be stopped for worker safety (CEDEX, 2009). 4.2 Breakwater type section The breakwater typical main section (Figure 10) comprised a seaward armor layer sloping at 2H/1V and a back layer at 1.5H/1V, composed of 150 and 50 t

Figure 9. Coruña outer port. Layout and general view

(Noya, 2008).

Figure 10. Coruña outer port breakwater. Type section

(Puertos del Estado, 2012).

blocks. It has a riprap core and three filter layers between the core and the main armor layer: 1 t quarry stone, and two 15 t blocks. The main armor layer is supported by a 3-5 t rubble mound berm at −28 m and the back armor layer by a 0.5 t rubble mound berm at −7 m, after which other 5 t rubble are used instead of blocks.

At the top of the breakwater, the main amour layer crest and the crown parapet lie at the same elevation (+25 m), so that this element is protected from the waves. The prototype breakwater toe was at −42 m, and the bottom in the model front was recreated using a ramp sloping at 1.5%, representative of the bathymetry.

4.3 Breakwater construction stages

Five construction stages were studied for (A) tempo rary protection work: 1 t core protection armour stone,

one of the two layers using150 t blocks, and several lay outs for the temporary crown protection using150 and 50 t blocks(Figure 11).

The following 7 stages were reproduced for (B) waves at which works must be stopped: 1. Core pro tected by a 1 t filter layer; 2. Core protected by one 15 t filter layer, levelled at +10 m; 3. Core protected by two 15 t filter layer, levelled at +10 m, 4; Core pro tected by one 1 t filter layer with crown berm; 5.Core protected by one 150 t block layer, levelled at +10m; 6. Core protected by two 15 t filter layer, levelled at +10 m and 7.Core protected by one 150 t block layer

with crown berm. Figure 12 shows 1; 4 & 7 stages. Figure 11. Temporary crown protections. Construction stages 1 to 5. Figure 12. Waves at work must be stopped. Construction stages 1; 4 and 7. Figure 13. Core grain size distribution. 4.4 Test waves For (A) temporary construction protections, the test were performed with Tp = 20 and Hs from 7 m up to the crown failure using a Jonswap spectrum (peak parameter Tp = 3.3). Each storm was created using successive sea conditions: Hs increasing, m by m, and four tide phases: low, mean (rising), high and mean (ebbing). The duration of each condition was set so that the number of waves generated was sufficient to ensure the section equilibrium state (610÷690). For (B) waves at which work must be stopped, also a Jonswap spectrum was also used (? = 3.3) withTp = 8, 10, 12, 16 y 21 s and Hs increasing from 1 m in 0,2 m steeps until reaching the stop criterion: “water must Table 1. Characteristics of the blocks and quarry stones. Prototype Model Weight Dens. Block side Weight Dens. Block Blocks (t) (t/m 3) (m) (g) (g/cm 3) side (cm) Main layer 150 2.4 3.97 6193 2.27 13.97 Back layer 50 2.4 2.75 2064 2.27 9.68 Filter 15 2.4 1.84 620 2.27 6.48 Quarry Weight Dens. Equiv. side Weight Dens. Equiv. side stone (t) (t/m 3) length (m) (g) (g/cm 3) length (cm) Core 3–5 2.6 1.15 100–170 2.6 3–4 Sea side filter 1 2.6 0.72 28–30 2.6 2.1–2.4 Back armour layer 5 2.6 1.24 148–180 2.6 3.98–5

Back filter 0.5 2.6 0.57 14–18 2.6 1.5–2

not reach to the work platform”. 500 waves were repro

duced at a rate $H_{max} = 1.9\ H_s$ and three sea levels:

3.0; 4.0 and 5.0 m.

4.5 Test flume

The facility used in this test was the Large Flume, introduced in section 3.5 for the Barcelona breakwater case.

4.6 Test characteristics

4.6.1 Similarity law, model scale and scale effects

As is general practice, Froude’s similarity was used to a geometric scale of 1:28.5. This was chosen considering: wave generation capacity to reproduce waves proposed; flume wave gauges operating range; model size and scale effects.

Scale effects due to forces of elasticity forces (Fe) and surface stress (Fσ) can be disregarded for the same reasons explained for the Barcelona breakwater. For Fμ it can also be disregarded is because the Reynolds model number (Re) m is > 3 × 104 for HD > 0.09 m, so the model flow is turbulent as in the prototype.

The effects of the core permeability, defined in the breakwater design by Dmax = 100 kg and 5% maxi

mum size D < 1 kg, are also negligible, because this element was modelled with sizes somewhat larger than those that there would be if the geometric scale were

applied [Dm = Dp/λ× K, where K = 3.4 for the small

sizes, Hughes (1993)]. The grain size distribution

curve used can be seen in Figure 13.

4.6.2 Test waves calibration

Before the tests, waves were calibrated to create paddle

movements that would make it possible to reproduce

wave characteristics adapted to the defined JONSWAP

spectrum. With a view to this, a record was generated

for each of the waves and water levels whose duration

was such that the number of waves was sufficient for

statistical analysis. Figure 14. Construction stages. Core, filter and first blocks armour layer. 4.6.3 Model materials The design prototype block density was γ p = 2.4 t/m 3 . To maintain the model/prototype density ratio, (γa) p /(γa) m = 1.025 must hold, so it was necessary to manufacture (γa) m = 2.34 g/cm 3 model blocks. Although blocks were constructed to reach that density, this was not achieved, which meant that a correction to the model/prototype stability number (Ns) (1) conservation was necessary (Hudson et al., 1979), to guarantee similarity between the two systems and the model elements weight is obtained by (2): Table 1. shows the characteristics of the blocks and the quarry stones (prototype and model).

4.6.4 Model construction

Construction began with the core, followed by the

filters: 1 t quarry stones and 2 layers of 15 t blocks man

ually placed, making sure that there were no smooth

zones, so the main armor layer could have a coarse

support (Figure 14).

When constructing the main armour layer (150 t) a placing pattern, using coordinates, was devised with 40 % porosity, as included in the project. The coordinates were established with a gap of 1.50 m between the blocks in each row and in such a way that they were in contact with the ones on the row below, arranged in staggered formation.

To find the armor layer characteristics, poros ity (p), packing density (φ) and placing density (d) were calculated [p = n ∘ blocks·block vol/armour vol.; φ = 1 − n ∘ layers·k p ·(1p); d = n ∘ blocks/surface armour; k p = 1.05]. The porosity turned out to be slightly less than the 0.40 design value and the packing density slightly above 1.20, normal value for blocks placed randomly.

4.7 Test methodology

4.7.1 Wave generation measurement

The waves generated were measured to make sure they were the same as the test waves proposed. Mea surements were analyzed with the Mansard-Funke method (Mansard and Funke, 1980), at 50 Hz with 3 gauges to calculate Hs incident and reflected, and the GEDAP application (National Research Council, Canadian Hydraulic Laboratory) being used.

4.7.2 Stability test. Damage criteria

For the temporary construction protections (A), the damage at each storm stage was not repaired and the following activities were performed: measuring the section incident/reflected waves, counting the elements displaced in the section zones and photographs and video at the start, during and at the end of each storm stage.

4.7.3 Overtopping measurements

The number of overtoppings were counted (in the previous test the volume was measured).

4.8 Test results

(A) Temporary construction protection

The best construction stage performance was for Stage 2. It failed at Hs = 11 m and Tp = 20 s (Figure 15). Others failures stages occurred at Hs = 9÷10 m.

(B) Wave height at which the works must stop for worker safety

First overtopping was for Hs = 1.35 m at Stage 1 with 5.0 m water level and Tp = 21 s and the highest overtopping was for Hs = 6.92 m, water level 3.0 m and Tp = 21 s. Figure 16 shows the Hs first overtopping for 5; 4 and 3 m water levels. Figure 15. Stage 2. Final situation (failure): Hs = 11 m, Tp = 20 s. Figure 16. Stage 1. Fist overtopping. Hs (m) versus Tp (s). 4.9 Breakwater construction behavior Table 2 shows the major storms withstood by the breakwater during its construction. Damage only affects stretches under construction and temporary protections, in a similar way to what happened in the physical model test, which can be seen in figure 17, shown by way of comparison. 5 CONCLUSIONS The conclusion of the two cases given in this paper and from others construction stages tests conducted in the CEDEX, can be summarized as follows: – Usefulness of physical models as a tool for studying the construction phases of port breakwaters – Application to both: rubble mound and vertical breakwaters – Possibility to analyze the construction phases of the type section (2D tests) as well as the construction progress as a whole (3D test) – In both cases the size of the models must be as large as possible, especially for the rubble mound breakwaters, considering that among its components flow should be turbulent, as actually occurs in the prototype, for which Reynolds number should be higher than 30,000.

Table 2. Major storms during construction.

Low-cost 3D mapping of turbulent flow surfaces

A. Nichols & M. Rubinato

Civil and Structural Engineering Department, The University of Sheffield, UK

ABSTRACT: This work presents a new measurement technique with potential to remotely measure turbulence

processes based on their linkages with the dynamic water surface. Previous work has demonstrated these linkages,

but accurate 3D measurement of turbulence-generated rough water surfaces is problematic. Microsoft Kinect

sensors have previously been used for measuring gravity waves in coloured (opaque) water. Here the sensor is

applied to clear water surfaces, and data is validated against a conductance-based wave probe. These data show

that the Kinect sensor can characterise relatively large gravity waves in clear water via the principle of refraction.

The same principle is able to capture the dominant components of turbulence-generated water surfaces, though

discrete features can be ambiguous. It is postulated that a flat bed would result in an unambiguous measurement.

Application of this technique to turbulent flows could lead

to new applications in flood studies, river monitoring, and sewer flow measurement.

1 INTRODUCTION

Recent work has shown that water surface fluctuations can be associated with the underlying velocity field and turbulence, which could in turn be related to the flow conditions, boundary conditions and hydraulic processes (Horoshenkov et al., 2013; Fujita et al., 2011). The ability to relate the underlying flow to the free surface has the potential to enable remote, non-intrusive measurement of flow processes. It could therefore find applications in flow monitoring, sedi ment entrainment, and mixing studies.

This potential can only be realised via an abil ity to accurately characterise the three-dimensional dynamic surface roughness patterns of turbulent flows. Several techniques exist for measuring water surface position, including: acoustic and microwave sensors (Boon et al.,2008); stereoscopic imaging (Tsubaki et al., 2005); infrared and laser displacements tech niques (Daida et al., 1995; Takamasa et al., 2000); laser induced fluorescence (Nichols, 2013; Lommer et al., 2001; Charogiannis et al., 2015); and submersible pressure transducers (Rubinato 2015). However, these techniques are limited in their spatial resolution, and generally measure only in one or two dimensions. This work presents an innovative low-cost method to measure surface water patterns by using a Microsoft Kinect sensor, which has the ability to measure at reasonable spatial resolutions and in three dimensions. This device was originally developed to measure and interpret human gestures as a video game input technology. The sensor employs a variant of image-based 3D reconstruction, by emitting a pattern of infra-red dots, and interpreting the detected pattern to determine the three dimensional surfaces in the field of view. The resulting output is an RGB image of 480 × 640 pixels, and a depth map which is also 480 × 640 in size, containing for each pixel a value representing the distance from the sensor. To date, Kinect sensors have been used to measure fluid surfaces in the form of gravity waves generated in coloured (opaque) water (Combes et al., 2011) and sand flows (Caviedes-Voullieme et al., 2014). They have never been used to measure turbulence-generated water surface roughness. They have also never been used to measure clear water surfaces because the infrared light generally passes through the water surface if it is not opaque, reflecting instead from the bed of the flow (if it is shallow enough). For all the tests conducted to date by other scientists, the water had been coloured with the addition of white dye to enhance the liquid's light diffusivity, enabling it to appear as a solid surface. This study examines the hypothesis that the infrared light is refracted as it passes through the air-water interface, and reflects from the bed, and that local changes in the gradient of the interface result in perceived changes in the position of the bed surface. A Kinect sensor is positioned above an experimental channel within the University of Sheffield hydraulics laboratory. Initially, large-scale gravity waves are generated on a range of six flow conditions and the resulting fluctuations in bed measurements are validated against data collected with a conductance-based wave probe in the same location. Subsequently, the smaller scale turbulence-generated roughness is measured by the Kinect sensor and the wave probe. In this manner, this paper aims to demonstrate that this technology has the potential to measure clear water surface waves, which could be applied to a range of applications where the addition of colourants may be undesirable or impractical (for example river

Figure 1. View of the flume from the downstream end (left),

and the inlet system equipped with point gauges (right).

flows). This also reduces the cost and complexity of

measurements.

2 METHODOLOGY

2.1 Flume setup

Testing was undertaken in a flume within the Univer

sity of Sheffield hydraulics laboratory. The behaviour

of the water surface was altered by adjusting the gen

eral flow conditions to generate a range of steady,

uniform shallow flows (from 1.8 l/s to 21.7 l/s) over

a rough, sediment boundary. The flume (Figure 1)

has an experimental length of 8 m, a width of 0.3 m

and depth of 0.3 m. For this set of tests the slope was

fixed at 0.001. The measurement section was located

at 3.50–4.00 m from the upstream end of the flume.

The inlet pipe to the flume is fitted with an elec

tronic control valve operated via Labview software.

Discharge through the channel can thereby be con

trolled, and is measured using a magnetic flow meter in

the supply pipe. Downstream of the experimental sec

tion the flume is fitted with an adjustable weir to enable uniform flow to be achieved. Six flow conditions were examined, ranging in depth from D = 0.03 m to 0.120 m, with mean velocity from U = 0.18 to 0.60 m/s. The uniform flow depth was measured with point gauges (accuracy ±1 mm). The flow conditions examined in this study are displayed in Table 1. For each flow condition, data was first recorded for gravity waves generated manually in the flow by submerging a paddle to the mid-depth and moving it back and forth in the streamwise direction. These waves were approx imately 20–30 mm in height. Then data was recorded for the raw water surface fluctuations generated by the flow turbulence.

2.2 Wave probe

To validate the surface measurement, a conductive wave probe was installed at the centre of the flume, 3.95 m from the upstream end of the flume. The probe consisted of two vertical 1 mm diameter conductors separated by 15 mm in the lateral direction (Figure 2).
Table 1. Flow conditions utilised for the full range of tests (1 to 6). Flow Velocity Water Test (m 3 /s) (m/s) depth (m) 1 0.0017 0.18 0.030 2 0.0049 0.34 0.048 3 0.0081 0.41 0.066 4 0.0116 0.46 0.084 5 0.0165 0.54 0.102 6 0.0217 0.60 0.120 Figure 2. Wave probe used to validate Kinect sensor results. A Churchill Wave Monitor unit was used to energise the probe and provide an output voltage proportional to water depth. The probe was calibrated by establishing flows of depth equal to those inTable 1 and displayed in Figure 3. The voltage output for each flow depth was recorded over a period of 30 sec, and averaged. The mean voltage for each flow depth forms the basis of the linear calibration, and was used to identify the first

order polynomial coefficients necessary to transform a recorded voltage into an instantaneous flow depth. 2.3 Acoustic doppler velocimetry (ADV) An Acoustic Doppler Velocimeter (Side-looking Nortek Vectrino) was used to record the threecomponent velocity at a rate of 100 Hz. The probe was positioned at 4.50 m from the flume inlet, at the centreline of the flume. For each flow condition, velocity was recorded at a height of 12 mm from the bed level and also at a height of y/d = 0.4. For the measurements taken at 12 mm from the bed level, tests 5 and 6 exhibited correlation values between 60–70%, while tests 1, 2, 3 and 4 were all above 80%. For the data collected at

Figure 3. Flow-depth conditions examined for wave probe

and Kinect measurements.

Figure 4. Mean velocity values measured at a height of

y/d = 0.4.

the height of y/d = 0.4, correlations for the full range

of tests were always above 75%. The majority of these

values are hence within the acceptable range defined

by Wahl (2000). The mean velocity values recorded

are displayed in Figure 4, where Vx is the streamwise

velocity,Vy is the lateral velocity, andVz is the vertical

velocity.

2.4 Kinect sensor setup

To spatially calibrate the Kinect sensor a chequerboard

pattern (see Figure 5) was placed on the flume bed

beneath it. The elevation of the grid was set to coincide

with each of the planned flow depths given in Table 1,

and images were recorded. A Matlab algorithm then

identified the vertices of the chequerboard, and used

these to determine a piecewise linear transformation which would map the Kinect images onto an orthog

onal Cartesian coordinate system. A calibration was

thereby calculated for each flow depth for the Kinect

sensor.

The depth map output by the Kinect sensor has arbi

trary units linearly proportional to the distance of an

image pixel from the sensor. In order to convert the

depth map into meaningful units, a depth calibration

must take place. A depth map was recorded for a flat Figure 5. Location of the Kinect above the area of study and an example of chequerboard seen from the Kinect sensor. Figure 6. Linear calibration of Kinect depth output. surface at different heights. This captured the change in sensor output for a known change in the position of the surface. For each pixel the linear relationship between sensor output and surface height was determined. This could then be used to transform the sensor output into a depth map in units of mm. Figure 6 shows an example of the linear calibration for the pixel location immediately between the two wires of the wave probe. The coefficient of determination for the linear fit was always greater than 0.96. For this study, the area around the wave probe was considered, in order to provide an initial validation of the Kinect sensing method. The wave probe was positioned 3.95 m from the flume inlet, and at the centreline. For the Kinect signal, the average surface elevation was taken for an area from 3.94 to 3.96 m in the streamwise direction, and extending 20 mm either side of the flume centreline. 3 RESULTS & DISCUSSION 3.1 Gravity wave data As described earlier, data was first recorded using the Kinect sensor and wave probe for gravity waves

Figure 7. Gravity waves measured by Kinect.

generated manually in the six flow conditions.A 30 sec

time series of data recorded on the Kinect sensor is

shown in Figure 7 for all six gravity wave cases. The

first thing to note is that the data represents the per ceived position of the flume bed. With no water, this would sensibly be zero; with water, the infra-red light is refracted toward the vertical and hence travels a slightly shorter path. This is why all the readings are above zero.The first three flow depths (30 mm, 48 mm, and 66 mm) show some progressive change; i.e. an increase in perceived bed level as a result of an increase in the water level. This also appears to result in pro gressively larger variation being detected. Beyond this depth, the overall level no longer increases, and the variation appears to decrease. This could be indicative of a limit being reached, beyond which the sensor no longer ‘sees’ the flume bed properly.

Since the Kinect sensor in this configuration mea sures perceived bed level rather than surface level, in order to compare directly between the Kinect and wave probe data, both data sets were reduced to a mean of zero, and the Kinect data was scaled so that its standard deviation matched that of the wave probe data. Figure 8 and Figure 9 show the direct compar ison between wave probe data (blue line) and Kinect depth recording (black line). In both cases a 3rd order low-pass Butterworth filter was used to remove high frequency spikes. This type of filter has been shown to be suitable for flow surface data (Horoshenkov et al., 2013). Figure 8 shows the result of a 5 Hz cutoff frequency in the filter, while Figure 9 shows a 1 Hz cutoff. It can be seen that generally the Kinect data matches the signal recorded on the wave probe. The agreement appears better for the data filtered to below 1 Hz. This is likely because the turbulence generated roughness has been removed. This finer scale rough ness is less likely to agree because of the different ways that the two systems measure their data (wave probe: average of two points 15 mm apart; Kinect: average of 20 mm × 40 mm area). The correlations between the two measurements for the 1 Hz filtered data are given in Table 2. It can be seen that for small-medium wave

Figure 8. Gravity waves filtered to below 5 Hz (Blue – wave probe; Black – Kinect). Figure 9. Gravity waves filtered to below 1 Hz (Blue – wave probe; Black – Kinect). Table 2. Correlation of gravity wave data. Water Correlation Test depth (m) coefficient 1 30 0.60 2 48 0.82 3 66 0.88 4 84 0.83 5 102 0.53 6 120 0.47 heights and flow depths, the Kinect-based measurements are reasonably accurate; but for higher depths and wave heights the error increases. 3.2 Turbulence-induced surface behavior The same process was then applied to the data recorded for turbulence driven surface roughness (much smaller scale). The six 30-second Kinect recordings are shown in Figure 10. Reassuringly, the same pattern

Figure 10. Turbulence generated surface fluctuations detected by Kinect sensor for 6 flows.

is observed as was seen in the gravity wave data. The average perceived depth increases with flow depth, up to 66 or 84 mm. Beyond this it is possible that the sen sor is unable to properly discern the bed position, as the mean level does not significantly change and the fluctuation reduces.

The 5 Hz filtered data in Figure 11 show very little agreement between the Kinect data and the wave probe, although some of the larger features do appear to relate.

The 1 Hz filtered data (Figure 12) sheds some light on the reason for this. Some of the traces, for example D = 30 mm and D = 84 mm show reasonably good agreement, while others only match certain individual features. This is reflected in the correlation values shown in Table 3. What is perhaps more interesting is that some of the features detected by the Kinect sensor appear to be the inverse of the corresponding features detected by the wave probe, for example between 7–11 sec for D = 48 mm. This can be explained by the fact that the fluctuating surface gradient is altering the position of the infra-red pattern on the rough gravel bed. For this fine-scale surface roughness, the movement of the infra-red pattern on the bed can be close to the grain scale. This means that a change in the surface gradient could cause the pixel location to move to a local peak of trough on the bed, causing ambiguity in the measurement. This was not the case for the larger gravity waves, which would refract the light to a larger degree, hence detecting broad perceived changes in bed level, rather than grain-scale fluctuations. It is therefore postulated that the turbulence-generated sur face roughness may be better detected if measuring over a flat bed (or section thereof).

3.3 Links with flow field

The overall aim of this area of work is to develop tools for remote monitoring of turbulence-driven pro cesses. While measurement of the surface fluctuations is known to be useful, a direct link has not been shown

Figure 11. Turbulence-generated waves filtered to below 5 Hz (Blue – wave probe; Black – Kinect). Figure 12. Turbulence-generated waves filtered to below 1 Hz (Blue – wave probe; Black – Kinect). Table 3. Correlation values for turbulence-driven surface patterns.

Test	Water depth (m)	Correlation coefficient
1	30	0.57
2	48	0.40
3	66	0.13
4	84	0.50
5	102	0.20
6	120	0.20

between the free surface fluctuations and the in-flow turbulence. Here, the ADV data was used to calculate the standard deviation in the three velocity components at a position of $y/d = 0.4$. This turbulence intensity is related to the measured wave height (standard deviation of wave probe recording) in Figure 13. It can be seen that the wave height generally increases with

Figure 13. Standard deviation of wave probe data as a func tion of the three-component velocity fluctuation at $y/D = 0.4$.

Figure 14. Standard deviation of Kinect data as a function of the three-component velocity fluctuation at $y/D = 0.4$.

increasing velocity fluctuation. At a flow depth of around 84-102 mm this relationship begins to break down. It is highly likely that beyond this point the depth to width ratio is no longer small, and hence shallow flow assumptions do not apply. In particular, secondary flows and three-dimensional flow patterns begin to form which can explain the mismatch between the wave probe and ADV fluctuation measurements.

What is of note, is that the standard deviation of the surface elevation time series recorded by the Kinect sensor (Figure 14) shows a similar trend for the first three data points: surface fluctuation increasing with velocity fluctuation. Unfortunately, as remarked upon earlier, for depths beyond 66 mm the Kinect sensor does not accurate resolve the scale of the surface waves, and therefore the relationship with velocity fluctuation is not upheld.

4 CONCLUSIONS

While Kinect sensors have previously been used to

Nezu, I. & H. Nakagawa. 1993. Turbulence in Open-Channel Flows. Rotterdam: Balkema.

Nichols, A. 2013. Free Surface Dynamics in Shallow Tur bulent Flows. Thesis (Ph.D.) University of Bradford, https://bradscholars.brad.ac.uk/handle/10454/7345.

Nichols,A.,Tait, S., Horoshenkov, K., & S. Shepherd. 2013.A non-invasive airborne wave monitor. Flow Measurement and Instrumentation 34: 118–126.

Roy, A.G., Buffin-Belanger, T., Lamarre, H. & A. Kirkbride.

2004. Size, shape and dynamics of large scale turbulent

flow structures in a gravel bed river. Journal of Fluid

Mechanics, 500: 1–27.

Rubinato, M. 2015. Physical Scale Modelling of Urban

Flood Systems. Thesis (Ph.D.) University of Sheffield,

http://etheses.whiterose.ac.uk/9270/. Takamasa, T. & T. Hazuku. 2000. Measuring interfacial waves on film flowing down a vertical plate wall in the entry region using laser focus displacement meters. International Journal of Heat and Mass Transfer, 43: 2807–2819. Tsubaki, R., & I. Fujita. 2005. Stereoscopic measurement of a fluctuating free surface with discontinuities. Measurement Science and Technology 16:1894–1902. Wahl, T.L. 2000. Analyzing ADV Data Using WinADV. In: Joint Conference on Water Tesources Engineering and Water Resources Planning & Management, Minneapolis, Minnesota: 1–10. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Sediment transport measurements in the Schelde-estuary: How do acoustic

backscatter, optical transmission and direct sampling compare?

S. Thant

Antea Group, Antwerp, Belgium

Y. Plancke & S. Claeys

Flanders Hydraulics Research, Antwerp, Belgium

ABSTRACT: Measuring sediment transport remains one of the most challenging aspects in river engineering.

In 2014, Flanders Hydraulics Research performed several field measurement campaigns in the Schelde-estuary.

The goal of these campaigns was to gather datasets for numerical model (Delft3D and Telemac-suites) validation

of sediment transport and to increase our system understanding. During these campaigns both optical and

acoustic measurement techniques were used. Differences in sediment concentration patterns were found between different techniques. In 2015 a specific field campaign was organized to calibrate both the acoustic backscatter signal (Nortek AquaDopp) and the optical transmission signal (LISST-100X), while collecting multiple water and sediment samples using pump samplers and a water trap. A good agreement was found between measured sediment concentration from the direct samplers and the acoustic backscatter and LISST-100X data. Although, differences can occur, most likely because of the sand-silt rate changes, the presence of flocs and the accuracy of the instruments.

1 INTRODUCTION AND BACKGROUND

Within several water engineering projects sediment transport plays a crucial role (e.g. morphological changes). State-of-the-art numerical models still have their limitations in predicting these evolutions, which can lead to uncertainties in results, and trigger the precautionary principle within environmental impact and appropriate assessment (within the scope of the European Bird and Habitat Directive). To improve the numerical models, but also our system under standing, good sediment transport and morphological measurements are crucial. Where multi-beam echo soundings result very precise topo-bathymetric infor mation, measuring sediment transport remains one of the most challenging aspects in river engineering.

In 2014, Flanders Hydraulics Research performed several field measurement campaigns in the Schelde estuary to gather datasets for numerical model (Delft3D and Telemac-suites) validation of sedi ment transport. During these measurement campaigns, direct (Delft bottle, pump sampling) and indirect (opti cal and acoustical backscatter) techniques were used. Each technique has its advantages and disadvantages (Gray et al., 2009): the optical backscatter (OBS) sensors used over longer periods (~4 weeks) were prone to bio-fouling (Figure 1), which disturbed the measurements.

Where simultaneous measurements of flow char acteristics using Acoustic Doppler Profiler were

performed, acoustic backscatter data was used to make
Figure 1. Bio-fouling on lower OBS-sensor. estimates of the sediment concentration after the biofouling occurred on the OBS-sensors. It was found that the patterns over a tidal cycle were not fully in agreement, which could be related to sensor sensitivity for different sediments (silt vs sand). Unfortunately, no specific field campaign was organized to calibrate the acoustic backscatter, so the results had large uncertainties. Several comparisons have been made in the past, both in situ and in laboratory conditions (Gartner et al., 2001; Meral, 2008). To gain a better insight in these aspects, specific field measurement campaigns were performed in 2015, using different techniques at one location over a limited period of one tidal cycle in the Schelde-estuary.This paper focusses on sediment concentration and grain size distribution measurements,

Figure 2. Measuring locations along the river Scheldt (field campaigns 2015). Measuring location Appelzak located

downstream of the Dutch_Belgian border, and locations Oosterweel and Ketelplaat, positioned more upstream.

and discusses how acoustic backscatter, optical trans

mission and direct sampling compare to each other.

Different direct and indirect measuring techniques for validating LISST-100X data are analyzed and their performance discussed, regarding to sediment concen tration and grain size distribution. A supplementary analysis was made to understand to which extent backscatter data of an AquaDopp can be accepted as a proxy for sediment concentration.

2 STUDY AREA

The Schelde-estuary has a length of 160 km and is located in Flanders (up-estuary, called “Zeeschelde”) and the Netherlands (down-estuary, called “Wester schelde”). The estuary is characterized by a macro tidal regime, ebb and flood currents, a longitudinal salinity gradient and important sediment transports (both sand and silt), leading to important morpholog ical changes.

Over the past centuries several human interfer ences have taken place in and along the estuary: starting with important poldering of areas along the estuary, dike-building, cutting-off of several bends, dredging works to guarantee the port accessibility and sand extraction for commercial reasons. Beside

these human activities sea level change occurred and has caused changes in the morphology of the estuary and thus the tidal penetration in the estuary. In 2001 Flanders and the Netherlands signed a memorandum of understanding in which a "Long Term Vision" (LTV) strategy and its objectives for the Scheldtestuary was defined. The LTV focusses on the three main functions of the estuary: (1) safety against flooding, (2) port accessibility and (3) nature. In 2010 an integrated monitoring programme (called "MONEOS") was agreed, which should make it possible to evaluate the effects of the different projects on the physical and ecological system (Plancke et al., 2012). Beside this "system"-monitoring, additional "project" and "research" monitoring takes place. Within the "research" monitoring, one of the projects focusses on the applicability of measurements techniques. This paper deals with specific field campaigns that were organized in 2015 to compare both the acoustic backscatter signal (Nortek AquaDopp) and the optical transmission signal (LISST-100X), while collecting multiple water and sediment samples using pump samplers, a Delft bottle and a water trap. Data was obtained at 3 measuring locations along the Schelde-estuary (Figure 2): Appelzak, located just down-estuary of the Dutch-Belgian border (KM 53 from the mouth), Ketelplaat (KM 65 from the mouth), located just upestuary of the Dutch-Belgian border, and Oosterweel (KM 75 from the mouth) more up-estuary.

3 MATERIALS AND METHODS

The following direct and indirect measuring instru

ments were used during the field campaigns of

12/08/2015 and 31/08/2015:

– LISST-100X : measurement of grain size distribu

tion and sediment concentration.

– Centrifugal pump : water sampling

– Water trap : water sampling (only during campaign

of 12/08/2015)

– Delft bottle : measurement of sediment transport

- AquaDopp : measurement of flow velocity and sediment concentration (by acoustic backscatter)

When using these indirect measurement techniques (e.g. LISST-100X), effects of bio-fouling and mud accumulation on the optical sensors, as the role of par ticle density (mineralogical or organic compounds) are to be considered during measurement analysis. Addi tionally, processes such as mechanical selection of particle grain sizes or e.g. the formation/destruction of flocs are subject to the type of direct measure ment technique, which are often used as calibration for indirect techniques.

3.1 LISST-100X (Sequoia)

The LISST-100X was mounted in a frame (Figure 3), attached to a winch at the rear end of the vessel (star board side). The frame was equipped with 2 side pan els, ensuring a transverse orientation of the device on the current and a complete flow-trough of transported sediments trough the measuring volume. Sampling frequency was 1 Hz. Total volume concentrations can be obtained by the sommation of the volume concen trations detected at each of the 32 detector rings of the LISST-100X. Taking into account the density of the sediment, mass-sediment concentrations can be derived. The 32 detector rings also provide informa tion on the granulometry of the in situ sediment by

which median grain sizes can be deduced.

3.2 Centrifugal pump and water trap

The centrifugal pump and water trap were used for

water sampling, by which a calibration of the LISST

100X and acoustic backscatter data was possible.

The opening of the hose of the centrifugal pump

was fixed at the frame, at the same heigth of the LISST

100X, in a way the water sampling did not influence the

measurements of the LISST-100X. Samples (1 liter)

were taken at regular intervals.

The water trap is a tube, horizontally positioned,

closed off by a drop weight once the desired measuring

depth is reached.

Samples were analyzed in the labortory for sed

iment concentration by using a standard filter of

0.45 µm. In some cases an additional distinction

between sand (>63 µm) and silt (<63 µm) was made

by the use of an extra filter of 63 µm. Figure 3. LISST-100X (left) and AquaDopp on the tail of the suspended Delft bottle (right) during field campaign. 3.3 Delft bottle During the campaign of 12/08/2015 the suspended Delft bottle was used. The sampling time was 10 min, after which the sediment load in the bottle was determined. On 31/08/2015, the Delft bottle was mounted on a frame, at 35 cm and 47 cm above the bottom. Sampling time was 3-5 minutes, after which the sediment load was defined. Sediment transport can be deduced of the in situ sediment volume, taking into account the sediment density and porosity. 3.4 AquaDopp (Nortek) At the campaign of 12/08/2015, the AquaDopp was mounted on the tail of the suspended Delft bottle (Figure 3), using the High

Resolution measuring mode (HR mode). During data analysis, anomalies in the data were detected and it was ascertained that the High Resolution configuration mode was not suited for these type of measurements. Additionally, it was found that the suspended Delft bottle was susceptible to tilting and pitching during measurements, leading to distubance of the acoustic signal. Although some data seemed to be usefull, it was decided to discard this data. At the campaign of 31/08/2015 the Normal configuration mode was used, while theAquadopps were fixed on the frames of the LISST-100X and Delft bottle. Besides the flow velocity (magnitude and direction), also the backscatter data was analyzed. 3.5 Mastersizer 2000 (Malvern) In situ samples (pump, water trap and Delft bottle) were transported to the laboratory where a granulometric analysis was performed using the Malvern Mastersizer 2000. This device uses laser diffraction to determine the granulometric properties of the samples. Before analysis, samples were filtered on a 2 mm filter, to remove coarser parts (e.g. peat or organic matter in Delft bottle sample).

Figure 4. Volume concentrations detected by the LISST-100X compared to sediment concentrations obtained by water trap

and centrifugal pump samples (Appelzak, 12/08/2015).

4 RESULTS AND DISCUSSION

4.1 Sediment concentration

In Figure 4 volume concentrations (LISST-100X) and sediment concentrations (Pump sampler and water trap) of the measuring locationAppelzak (12/08/2015) can be observed. Results of water samples obtained by the centrifugal pump and water trap are almost identical, rejecting the hypothesis of a possible higher abundance in fine grained particles in centrifugal pump samples, due to pumping up the water. A high resemblance between the sediment concentration pat tern of the water samples and the LISST-100X can be observed. In both cases initial concentrations are low, both increasing around HW-90’ towards an abso lute maximum in sediment concentration at HW-75’. After this peak moment, sediment concentrations grad ually decline again. Also at Ketelplaat (31/08/2015) there is a good agreement between the patterns in sedi ment concentration of the LISST-100X and centrifugal pump samples. Higher values are observed between HW-90’ and HW-60’ (250 mg/l), rapidly declining afterwards (100 mg/l). Only at the start of the measure ments there is a discrepancy between both techniques, characterised by higher values detected by the LISST 100X compared to the measurements of the water sample.

At Oosterweel (31/08/2015) pump samples show a gradual decrease in sediment concentrations, con trary to the volume concentrations detected by the LISST-100X indicating an increasing trend (Figure 7). The latter being suspicious since current velocities, and subsequently also sedimenttransport capacity, are decreasing in this phase. 4.2 Grain size distribution At Appelzak significant differences in median grain sizes (d 50) were observed between LISST-100X measurements (80–120 µm) and the different pump and water trap samples (15 µm) and Delft bottle (40– 160 µm). For particular periods, values of the LISST100X and Delft bottle samples are similar. However, the sampling technique of the Delft bottle implies a selection of particle grain size (only sand is trapped in the bottle, fines (<63 µm) will pass through), causing the comparison between d 50 of LISST-100X data and Delft bottle data to be unjustified. Median grain sizes of pump samples at Oosterweel and Ketelplaat (Figure 5), are similar for both locations, but considerably lower compared to the d 50 values of the LISST-100X (100 µm (Oosterweel) to 180 µm (Ketelplaat)). Additionally, Figure 6 gives an overview of the full grain size distribution analysis for both LISST-100X and pump sample data at Appelzak (12/08/2015) and Ketelplaat (31/08/2016) around 1 hour before high water. As discussed above the median grain sizes of the Delft bottle samples (Oosterweel) are not representative regarding the measurements of the LISST-100X. The higher median grain sizes of the LISST-100X (Figure 6) can originate from the initial presence of sediment flocs in the system. These flocs brake up by pumping up the water, during transport or by handling of the water samples in the lab, leading to a higher concentration of fine grained particles and lower median grain sizes. Wren et al. (2000) confirm sample collection and handling can alter grain size distributions by breaking up aggregates.

Figure 5. Median grain sizes LISST-100X, Delft bottle and pump samples with (d 50) and without (d 50 _zonderUS) ultrasonic

treatment of the sample (Ketelplaat, 31/08/2015).

Figure 6. Comparison of grain size analysis for both pump sample and LISST-100X data at Appelzak (green) and Ketelplaat

(orange) at peak velocities around 1 h before high water.

Since lab conditioned measurements by the LISST

100X on calibrated sand (105 µm) also showed an

overestimation of the expected median grain size

(130 µm–150 µm), it is not clear how much of the

deviation in d 50 between LISST-100X and water sam

ples is due to measurement configuration or effective

change in sediment properties.

4.3 Tides and currents

The two AquaDopps used during the campaign of

31/08/2015, mounted on the frame of the LISST-100X

and Delft bottle, provide similar patterns in current

velocity for Oosterweel. At Ketelplaat there is a devi

ation between the two instruments, with the velocity data of the AquaDopp mounted on the LISST-100X frame, following the normal expected pattern (maximum velocity at 40 min before high water). This pattern is not observed in the data provided by the other AquaDopp (Delft bottle frame), where the missing velocity maximum before high water can be caused by an increased close-to-the-bottom sediment transport, leading to an attenuation of the acoustic signal. 4.4 Backscatter data (AquaDopp) Patterns of acoustic backscatter (AquaDopp mounted at LISST-100X frame) and sediment concentration data of pump samples are similar for both locations, Oosterweel and Ketelplaat. The backscatter data at

Figure 7. Acoustic backscatter data (AquaDopp), volume concentration (LISST-100X) and sediment concentrations

(pump sample) at Oosterweel.

Figure 8. Acoustic backscatter data (AquaDopp), volume concentration (LISST-100X) and sediment concentrations

(pump sample) at Ketelplaat.

Oosterweel confirms the decline in sediment concen

tration measured by the pump samples, contrary to the

LISST-100X data which, as mentioned earlier, showed

an increase in volume concentration (Figure 7). Meanwhile at Ketelplaat, backscatter data validates the sediment concentrations of both LISST-100X and pump sample data (Figure 8). These confirmations justify the general use of backscatter data as a

proxy for sediment concentration. Moreover, out of

range particles (for LISST-100X) can affect grain size

distributions (“rising tails”) and measured sediment

concentrations (Czuba et al., 2014; Agrawal & Pott smith, 2000). Also in these cases backscatter data can act as a useful validator for the LISST-100X data.

4.5 Calibration

LISST-100X volume concentration data andAquaDopp acoustic backscatter data provide an indirect estima tion of sediment concentration. The original values are converted to sediment concentrations based on a callibration curve, obtained by expressing the pump samples in function of the indirect signal (LISST 100X/Backscatter AquaDopp).

Using the water samples of the locations Oosterweel and Ketelplaat moderate correlations of $R^2 = 0.48$ and $R^2 = 0.64$ for repectively LISST-100X and backscat ter data were found. Making a distinction between the two measuring locations provides stronger corre lations for both techniques. A correlation of $R^2 = 0.92$ for LISST-backscatter data at Oosterweel was found, however, the relation found is inversely proportional, which is unlogical. The observed correlations, when making a distiction between sand and silt, are for both techniques moderate to weak. Both techniques though seem to have a higher sensitivity for variation in silt concentration.

The low correlation between the pump samples and the indirect measurements can be due to the altering of the sediment samples during effective sampling, transport and handling of the samples in the lab (e.g. destruction of flocs, formation of new flocs, …). Furthermore, measuring methods of the indirect tech niques (LISST-100X, AquaDopp) are assumed to be suitable for the terrain conditions present. However, because of the high variability in transported sedi ment (sand, silt, flocs, shape, …), both the optical and acoustic measuring technique will have limitations, ensuring indirect techniques to be an estimation of the real sediment concentration.

5 CONCLUSIONS

In 2015 specific field campaigns were organized to compare both the acoustic backscatter signal (Nortek AquaDopp) and the optical transmission sig nal (LISST-100X), with water samples collected using pump samplers, a Delft bottle and a water trap. Mea surements were carried out at 3 measuring locations along the Schelde-estuary, near the Belgian–Dutch border, where sediment transport takes places of both fine sand and silt.

Results show a good agreement between volume concentrations of the LISST-100X compared to sed iment concentrations determined by filtration of the pump samples and water trap samples. Only at Oost erweel an opposite trend can be observed between

volume concentration (LISST-100X) and sediment concentration data (pump samples). Backscatter data (AquaDopp) confirms the sediment concentrations obtained by the pump samples, both at Oosterweel and Ketelplaat, justifying the general use of backscatter data as a proxy for sediment concentration. Correlations between pump samples (Oosterweel and Ketelplaat) and LISST-100X and backscatter data were moderate, becoming stronger when making a distinction in measurement location. At Oosterweel the relation of the LISST-100X data is inversely proportional, which is unlogical. Distinctions in sediment type (sand/silt) gave moderate to weak correlations. Low correlations between pump samples and indirect measurements can be caused by the altering of the sediment samples during field sampling, transport and handling of the samples in the lab (e.g. destruction of flocs, formation of new flocs, …). Furthermore, it is assumed measuring methods of the indirect techniques (LISST-100X, AquaDopp) to be suitable for the terrain conditions present. However, because of a high variability in transported sediment (sand, silt, flocs, shape, …), both the optical and acoustic measuring technique will have limitations, ensuring indirect techniques to be only an estimation of the real sediment concentration. Median grain sizes (d 50) measured by the LISST100X are significantly higher for all locations than the corresponding d 50 values obtained by analysis of the water samples (pump samples, water trap samples, Delft bottle samples). This deviation can originate from the initial presence of sediment flocs in the system, which are broken up by pumping up the water, during transport or by handling of the water samples in the lab, leading to a higher concentration of fine grained particles and lower median grain sizes. Although, lab conditioned measurements by the LISST-100X on calibrated sand (105 µm) also showed an overestimation of the expected median grain size (130 µm–150 µm), it is thus not clear how much of the deviation in d 50 between LISST-100X and water samples is due to measurement configuration or effective change in sediment properties. Overand underestimations of median grain sizes by the LISST are also encountered in other studies, e.g. Filippa et al., 2011. Based on the analysis of the data of the measuring campaigns 2015, it is concluded more research, preferably under controlled conditions, is essential to determine the applicability and accuracy of the LISST-100X. 6 RECOMMENDATIONS To determine the accuracy of the LISST-100X regarding particle size, extra tests, under controlled conditions and with cohesive sediment, are proposed. The sediment sample, broken up, will be analyzed by the LISST-100X, Mastersizer and microscopy to test the accuracy of the LISST-100X for fine grained sediments. When adding a flocculation agent, also an analysis on flocs can be performed. As Rai & Kumar (2015) already proposed, also the effect of out of range particles on the LISST-100X data should be thoroughly tested.

Furthermore, the performance of the LISST-100X regarding sediment mixtures can be tested by creating a sediment mixture consisting out of e.g. 2 or more calibrated sediment fractions. Also in situ sediments can be used for the sediment mixture.

To obtain a thorough understanding of sediment characteristics in situ, a visual analyses can be per formed by the use of the INSSEV (in situ settling velocity meter). The INSSEV technique was already applied in a project concerning density currents in the Beneden-Zeeschelde. These images can give a conclu sive answer on the nature of the in situ sediments.

Agrawal, Y.C., Pottsmith, H.C. (2000). Instruments for par ticle size and settling velocity observations in sediment transport. Marine Geology. 168, 89–114.

Czuba, J.A., Straub, T.D., Curran, C.A., Landers, M.N., Domanski, M.M. (2014). Comparison of fluvial suspended sediment concentrations and particle-size distributions measured with in-stream laser diffraction and in phys ical samples. Water resources Research. 51, 320–340,

10.1002/2014WR015697. Filippa, L., Freire, L., Trento, A., Álvarez, A.M., Gallo, M., Vinzón, S. (2011). Laboratory evaluation of two LISST25X using river sediments. Sedimentary Geology. 238, 268–276. Gartner, J.W., Cheng, R.T., Wang, P., Richter, K. (2001). Laboratory and field evaluations for the LISST-100 instrument for suspended particle size determinations. Marine Geology. 175, 199–219. Gray, J. R., Gartner, J.W. (2009). Technological advances in suspended-sediment surrogate monitoring, Water Resources Research. 45, W00D29, doi:10.1029/2008 WR007063. Meral, R. (2008). Laboratory Evaluation of Acoustic Backscatter and LISST Methods for Measurements of Suspended Sediments. Sensors. 8, 979–993. Plancke,Y., Vanlierde, E., Taverniers, E., Mostaert, F. (2012). Monitoring of physical parameters within the scope of the Dutch-Flemish integrated monitoring program. (2012). Hydraulic Measurements and Experimental Methods 2012 Conference (HMEM 2012), Snowbird, Utah, August 12–15, 2012. pp. [1–6]. Rai, A.Kr., Kumar, A. (2015). Continuous measurement of suspended sediment concentration : Technological advancement and future outlook. Measurement. 76, 209–227. Wren, D.G., Barkdoll, B.D., Member, ASCE, Kuhnle, R.A., Member, ASCE, Derrow, R.W. (2000). Field Techniques for Suspendid-Sediment Measurement. Journal of Hydraulic Engineering. 126 (2), 97–104. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Field-deployable particle image velocimetry with a consumer-grade digital camera applicable for shallow flows

K. Koca, A. Lorke & C. Noss

Institute for Environmental Sciences, University of Koblenz-Landau, Germany

ABSTRACT

Detailed flow structure measurements in natural flows can provide new insights into the understanding of environmental fluid mechanics processes. Inspired by the need for flow detailization in natural flows,

a number of researchers (Liao et al., 2009; among others) employed the well-established Particle Image Velocimetry (PIV) technique, in field-based studies. PIV is advantageous over typical measurement tech niques (e.g. Acoustic Doppler Velocimetry) because it provides direct measurements of instantaneous veloc ity field, its spatial derivatives and spatial covariances in two (or three) spatial dimensions. These features allow for the calculation of instantaneous vorticity, dissipation rates (Westerweel et al., 2013), and turbu lent wave number spectra without relying on Taylor's frozen turbulence approximation, as well as observa tion of coherent flow structures.The field-PIV systems developed so far, however, utilized high power, sophis ticated laser systems, as well as sophisticated cameras, which makes them expensive and requires extensive deployment effort. We describe an alternative and inexpensive field-deployable PIV system based on a consumer-grade camera (GoPro Hero 4, GoPro Inc., USA) and a 225 mW (532 nm), continuous-wave laser module (Hercules, LaserGlow, Canada), which can be deployed in very shallow flows with a minimum water depth of 6 cm. To validate the developed system, simultaneous velocity measurements were performed in a flume using a Vectrino Profiler (Nortek, AS). The flow depth was constant at 30 cm, while the mean flow velocity, U, varied between 1.5 cm s^{-1} and 37.3 cm s^{-1} (four runs). Good agreement was found in a direct comparison of velocity time series (Figure 1). The velocities measured by the PIV were observed to be slightly less than those measured by Vectrino Profiler. The differences between the mean longitudinal veloc ities varied between 0.2% and 6.9%, with the biggest difference observed for U = 37.3 cm s^{-1}, whereas the differences were between 0.7% and 5.5% for the mean vertical velocities.

The root-mean-square velocity fluctuations mea sured by both instruments were in a reasonable agree

Numerical modelling of meandering jets in shallow rectangular reservoir using two different turbulent closures

Y. Peltier

University of Liege, ArGEnCo Department, Research Group of Hydraulics in Environmental and Civil Engineering,

Liege, Belgium

Université Paris-Est, Laboratory for Hydraulics Saint-Venant (ENPC, EDF R&D, CEREMA), Chatou, France

S. Erpicum, P. Archambeau, M. Pirotton & B. Dewals

University of Liege, ArGEnCo Department, Research Group of Hydraulics in Environmental and Civil Engineering,

Liege, Belgium

ABSTRACT: In this article, the shallow water equations are used to model meandering flows in shallow rectangular reservoir.Two distinct meandering flows in term of friction regime were modelled (frictional and non frictional) and two turbulent closures were tested with various values for their tuning parameters. One turbulent closure accounts for both the 2D horizontal ($k - \varepsilon$) and 3D vertical (algebraic model) turbulent mixing, while the second turbulent closure accounts for the unresolved scales of the shallow water equations through a subgrid scale model (Smagorinsky). The purpose here is to give advices on which turbulent closure is the most likely appropriate for modelling meandering flows. A Proper Orthogonal Decomposition (POD) is performed on the simulation results for extracting objective parameters accounting for transient behaviors and to be compared with experiments. Comparisons between models and with experiments indicate a great dependence of the simulation results to the modelling of the small scales of the turbulence.

1 INTRODUCTION

Shallow rectangular reservoirs are commonly used in natural and constructed environments. They are used for storing water (flood management, hydro-power generation) or for trapping pollutants and/or sedi ments (fresh-water production, stormwater treatment, protection of irrigation systems).

The optimal exploitation of such structures requires the control of the sediment dynamics. For an optimal storage of water, the sediment deposits must be mini mized, while they are maximized in settling reservoirs.

This control can only be achieved with the detailed knowledge of the flow developing within the struc ture (Dewals et al., 2008; Kantoush, 2008; Dufresne et al., 2010a, 2010b, 2012; Camnasio et al., 2011, 2013; Peltier et al., 2013, 2014a). Even for the simplest geometry of shallow reservoir, the flow has complex features such as regions of distinct mean velocities and large-scale horizontal coherent structures of various length-scales.

Lastly, Peltier et al. (2014a), proposed a classifica tion into four regimes of the flow developing in shallow rectangular reservoirs. This classification depends on the Froude number defined at the reservoir inlet, F , and on the geometry of the reservoir, characterized by the shape factor defined by Dufresne et al. (2010b)

as SF = L/▯B 0.6 b 0.4 (L = the reservoir length, b = the width of the inlet channel and ▯B = the width of the sudden expansion): – Symmetric (F < 0.21 and SF < 6.2); – Meandering (F > 0.21 and SF < 6.2); – Asymmetric (SF > 6.8) – Unstable (F > 0.1 – 0.15 and 6.2 < SF < 6.8). If the hydraulic conditions and the geometrical configurations for symmetric and asymmetric flows are well documented in the literature and numerical models have proved their ability at reproducing such flow features with or without sediments (Stovin and Saul, 2000; Dewals et al., 2008; Dufresne et al., 2011; Peng et al., 2011; Camnasio et al., 2013; Khan et al., 2013), the meandering and unstable regimes must still be investigated. Before trying to understand the mechanisms of generation of the unstable regime, it is necessary to go through the physics of the meandering jets. Recently, some studies have highlighted the mechanisms of the meandering flows in shallow rectangular reservoir. Using a Proper Orthogonal Decomposition, Peltier et al. (2014b) emphasized that the meandering jet is the result of at least two convective instabilities, which generate large-scale energetic turbulent structures within the flow. These large-scale turbulent structures contribute to increase the lateral momentum transfer between the jet and the rest of the flow and in

presence of sediments in the jet, increase their spread ing into the reservoir (Peltier et al., 2013). In another study, the ability of the shallow water equations to model meandering flows was investigated (Peltier et al., 2015). A $k - \varepsilon$ model coupled to an algebraic model for accounting for both the horizontal and verti cal turbulent mixing was used. This study emphasized that by adjusting the roughness height of the bottom in the friction modelling, the shallow water equations were able to model the dominant modes (in the POD sense) of the flow.

In the present article, we investigate the ability of the numerical model WOLF 2D (Dewals et al., 2008), which solves the shallow water equations, to repro duce meandering jets in shallow rectangular reservoir with two different types of turbulent closures and for a given roughness height. Two of the four flows detailed in Peltier et al. (2014b) were numerically mod elled using WOLF. The first tested turbulent closure is classical and accounts for two different length-scales: the 2D horizontal turbulent mixing is modelled by two additional transport equations ($k - \varepsilon$), while the 3D vertical turbulent mixing is treated with an alge braic model (Elder). The second turbulent closure is a subgrid-scale model (Smagorinsky), which models the unresolved scales of the shallow water equations (the small scales), while the large eddies are modelled through the shallow water equations. As meander ing flows are unsteady flows, the comparison of the experiments with the numerical modelling could not be performed using classical descriptors like velocity fields or vorticity fields. Experiments and numerical modelling were therefore compared through the use of the results of the Proper Orthogonal Decomposition of the fluctuating velocity fields. The eigenvalues, tem poral coefficients and spatial modes of the POD enable to plainly describe the nature of a flow.

2 MATERIAL AND METHOD

2.1 Experimental setup

The experiments modelled in this article were carried out in a flume based in the laboratory of engineering hydraulics of the University of Liege (ULg), Belgium. The experimental device is illustrated in Figure 1 (A complete description of the experimental setup is given in Peltier et al. (2014a, 2014b). The inlet channel is 2 m long and the outlet channel is 1.50 m long, the width of both channels, b, being equal to 0.08 m. The reservoir length, L, is equal to 1 m and the width of the sudden expansion, ∆B, is equal to 0.45 m. The bottom

was made of polyvinyl chloride (PVC) and the vertical

are made of glass. Given the roughness state of the

surfaces within the reservoir, the roughness height, k_s,

was evaluated to be equal to $k_s = 0.1$ mm.

In each experiment, the discharge, Q, was regu

lated (δQ = 0.025 L/s) and considered as constant.The

water depth, H , was measured in three positions in the
Figure 1. Sketches of the experimental flume and simulation domain (Peltier et al., 2014c, 2015). Table 1. Main characteristics of the measured flows.

ID	Q (L/s)	H (cm)	F (-)	S (-)	Re (-)
F	0.25	1.80	0.41	0.10	8,456
NF	1.00	4.20	0.46	0.03	24,2675

reservoir using an ultrasonic probe and was constant to the uncertainty (δH/H = 1%). In the present paper, x, y and z are the longitudinal, the lateral and the vertical directions of the Cartesian reference frame attached to the flume; x = 0 immediately downstream from the inlet channel and y = 0 at the right bank of the reservoir. z = 0 at the bottom of the reservoir. As most of the energetic coherent structures occurred in a horizontal plane and given the shallowness of the experiments, we assumed that the behavior of the flow at the surface was representative of the flow dynamics (Peltier et al., 2014b). The surface dynamics was therefore assessed using the surface velocity fields measured by Large-Scale PIV at a frequency of 25 Hz during 6’30”. The complete processing is detailed in Peltier et al. (2014a, 2014b). It should be noticed that given the low water depth in the experiments and the relatively smooth bottom, we considered that the velocity at the free surface was close to the depth-averaged velocity in most parts of the reservoir and we decided to not multiply by a surface coefficient (Le Coz et al., 2012) these velocities to be compared with the simulated depth-averaged velocities. Two very characteristic cases were selected amongst the existing dataset. The discharges, depths and the corresponding Froude number, $F = U_{in} / \sqrt{gH}$ (U_{in} = the mean velocity at the inlet and g the gravity acceleration), friction number (Chu et al., 2004), $S = f\,\Delta B/8H$ (f = the Darcy-Weisbach coefficient), and Reynolds number, $Re = U_{in} D/\nu$ (D = the hydraulic diameter of the inlet channel and ν= the kinematic viscosity of the water at 20 ∘ C) are summarized in Table 1.

The Froude numbers of the experiments are almost the same and are a direct consequence of the layout of the reservoir. In contrast, the friction number largely varies with increasing both the water depth and the discharge. Referring to the work of Chu et al. (2004) and confirmed by (Peltier et al., 2014b), the flow-case F belongs to the frictional regime, while NF belongs to the non-frictional regime. In the frictional regime, the sizes of the coherent structures developing in the flow are mainly controlled by the friction and the bottom generated turbulence: they are of the same order of magnitude as the water depth. In the non-frictional regime, the coherent structures are mainly controlled by the horizontal geometry and they are of the same order of magnitude as the sudden expansion �B.

2.2 Numerical modelling

The numerical modelling was performed using the aca demic model WOLF 2D developed at the University of Liege, which solves the shallow-water equations (Dewals et al., 2008). This finite volume model is robust and has proven its efficiency when dealing with flows in shallow reservoir (Dewals et al., 2008; Dufresne et al., 2011; Camnasio et al., 2012, 2013; Peltier et al., 2013, 2015). WOLF 2D includes a mesh generator and deals with multi-block Cartesian grids. This feature increases the size of possible simulation domains and enables local mesh refinements close to interesting areas, while preserving lower compu tational cost required by Cartesian grids compared to unstructured grids.

A linear reconstruction at cells interfaces, combined with a slope limiter is used to guarantee a second order space discretization. The convective fluxes are computed by a Flux Vector Splitting (FVS) method (Erpicum et al., 2010), which is Froude-independent and facilitates a satisfactory adequacy with the dis cretization of the bottom slope term.

The time integration is performed by means of a 3-step third-order accurate Runge-Kutta algorithm, limiting the numerical dissipation in time that could prevent the generation of eddies in the jet. The time step is adaptive, but for stability reasons, it was constrained by the Courant-Friedrichs-Lewy (CFL) condition based on gravity waves.

The simulations were conducted using one type of bottom condition deduced from experiments and previous simulations (Peltier et al., 2015) and char acterized by the roughness height k s = 0.1 mm. The side walls were considered as smooth. The Colebrook

White formula was used for modelling the friction (Dufresne et al., 2011; Camnasio et al., 2013).

Two types of turbulent closures were evaluated in the present paper:

– A Smagorinsky model (Hervouet, 2003) was first used for accounting for the small eddies that are unresolved by the shallow water equations. It is based on the mixing-length concept and belongs to the sub-grid models' group, i.e. only the eddies that are smaller than $\alpha \Delta x \Delta y$ (Δx and Δy being the dimensions of a mesh and α a tuning coefficient generally close to 0.1–0.2 (Rodi, 1980) are modelled, the larger eddies directly appearing in the solutions of the shallow water equations. In our simulations, the tuning coefficient α was set equal to 0.2. – A depth-averaged k-ε model with two different length-scales accounting for the 3D vertical and 2D horizontal turbulent mixing was then applied (Babarutsi and Chu, 1998; Erpicum et al., 2009). The 2D horizontal turbulent mixing is associated to transverse large-scale coherent structures and is modelled by two additional transport equations (k and ε), while the 3D vertical turbulent mixing accounting for the bottom-generated turbulence, which length-scale is of the same order as the water depth H, is treated with an algebraic model (Elder formula: $\nu_{t3D} = \lambda H u_*$, u_* = the friction velocity and λ= a generally taken equal to 0.08, but that can reach 2 in natural streamflow (Wark et al., 1990)). The $k - \varepsilon$ coefficients were the same as for unconfined three-dimensional flow, while three different λwere tested: 0 (means no 3D turbulence), 0.08 (the classical coefficient, see in Erpicum et al. (2009)) and 0.2 (high rate of 3D turbulence). The turbulent fluxes were evaluated by means of a centered scheme. In the simulations, the inlet channel of 2 m (Figure 1) was reproduced in order to have an injection in the reservoir as close as possible of the experimental configuration. The discharge read on the flow-meter was used as inflow condition at the beginning of the inlet channel, but the discharge was not uniformly distributed between the inflow cells. As prescribed by Dewals et al., a slightly disturbed discharge per cell distribution was indeed used in order to introduce a seed for asymmetry. The disturbance linearly varied from −1% to

1% across the inflow cells. The water depth measured 13 cm downstream from the outlet of the reservoir was prescribed as downstream boundary condition. For each simulation, the initial condition and the boundary conditions were obtained from an unsteady computation performed until a steady-state was reached (i.e. stable temporal and spatial oscillations of the jet). Once the steady state was reached, the results were recorded at a rate of 25 Hz in order to be compared with experiments (rate of the video-camera used for the LSPIV calculation). The grid spacing was set to 0.01 m and with a CFL value of 0.2, the time step was thus between 1.5×10^{-3} s and 4×10^{-3} s, the resulting sampling frequency being actually equal to 25 Hz ± 2.3 Hz. The choice of the grid spacing was a compromise and was guided by the grid independence tests presented in Camnasio et al. (2012) and Dufresne et al. (2011), which are based on the grid convergence index (Roache, 1997). Dufresne et al. (2011) highlighted the great sensitivity of reattaching flows to the grid fineness, but Camnasio et al. (2012) showed that the characteristic frequency of the meandering jet is weakly affected.

Table 2. Main characteristics of the simulated flows (with

$k_s = 0.1$ mm).

ID	Turbulent closure	λ (-) | α (-)
F-KE	$k-\varepsilon$	$\lambda= 0.08$
F-KE0	$k-\varepsilon$	$\lambda= 0$
F-KE0.2	$k-\varepsilon$	$\lambda= 0.2$
F-SM	Smagorinsky	$\alpha= 0.2$
NF-KE	$k-\varepsilon$	$\lambda= 0.08$
NF-KE0	$k-\varepsilon$	$\lambda= 0$
NF-KE0.2	$k-\varepsilon$	$\lambda= 0.2$
NF-SM	Smagorinsky	$\alpha= 0.2$

Figure 2. Two instantaneous surface velocity field of flow

case NF.

The different simulations are resumed in Table 2.

2.3 Flow characterization

The shape of the meandering jets is the consequence of at least two convective instabilities (Peltier et al., 2014b). The main instability is sinuous and is responsible for the meandering of the jet. It is made of an alternative succession of counter-rotating structures along the reservoir centerline. The second instability is varicose and is responsible for the lateral growth of the jet with increasing longitudinal distance in the reservoir. This instability is made of counter-rotating structures on both sides of the reservoir centerline. The description of the dynamics of those structures and of the unsteadiness of the flow (Figure 2) cannot be performed with the usual flow descriptors (velocity fields, Reynolds shear stresses). A Proper Orthogonal Decomposition was therefore used (Holmes et al., 2012). This method gives access to new flow descriptors by decomposing a collection of N snapshots of the fluctuating horizontal velocity fields, $u'(x_p, y_p, t_n) = u'(x, t)$ $(n, p \in \mathbb{N}^*$, $t = t_{n+1} - t_n = CST$ and $x \in \Omega \subset \mathbb{R}^2)$, which are square integrable functions (i.e. $u'(x, t) \in L^2(\Omega)$), into an orthonormal basis of M spatial functions $\varphi_m(x)$ of $L^2(\Omega)$, called spatial modes, and an orthogonal basis of M temporal coefficients, a m (t)(m ∈ {1, . . . , M ≤ N }

and M , N ∈N ∗) , such that: || || L 2 being the induced norm in L 2 (▯) (i.e. the root mean square of the inner product for L 2 (▯)). The snapshot method applied in this paper proceeds in three steps. (i) The temporal correlation matrix C is first calculated: C ∈R N×N and W ∈R N×N , W = a diagonal weighting matrix, for which the elements along the diagonal are the cell volumes of each of the P grid points of one snapshot. (ii) The temporal coefficients a m (t) are then obtained from the resolution of the eigenvalue problem defined as: As C is definite, positive and symmetric, the eigenvalues λ m are all real with λ 1 ≥ λ 2 ≥ . . . ≥ λ N > 0, and the eigenvectors α m (t) are orthonormal. The temporal coefficients, a m (t), are function of the eigenvectors and of the eigenvalues and they are orthogonal: with ⟨a n ⟩ = 0 and ⟨a n a m ⟩ = λ n δ nm . (iii) The spatial modes are finally obtained by projecting the fluctuating velocity ensemble onto the temporal coefficients, i.e.: with ‖φ m ‖ 2 L 2 =φ T m Wφ m = 1 (i.e. the spatial modes are orthonormal with respect to the inner product in L 2 , φ T m Wφ m). 3 RESULTS AND DISCUSSION The POD was applied on 9,000 experimental and numerical fluctuating velocity fields for each flowcase resumed in Table 1 and Table 2, the size of the computation grid being equal to ~10,000 points. Thanks to this decomposition, the modes (i.e. the coherent structures) contributing the most to the flow were identified. 3.1 Energy As previously highlighted, the POD allows the identification of the flow structures that contribute the most to the flow energy (Brevis and García-Villalba, 2011) and sorts the modes by descending energy (Eq. 3).This decomposition is optimal in average on the time interval covered by the data (Couplet et al., 2003), i.e. there

Figure 3. Mean fluctuating kinetic energy contained in the

mth modes and normalized by the square of the velocity at

the inlet.

is no better decomposition for discriminating the struc

tures with respect to their respective energy (Perrin

et al., 2007; Cavar and Meyer, 2012).

The mean fluctuating kinetic energy in the mth

mode and normalized by the square of the velocity at the inlet is displayed in Figure 3 for experiments and numerical simulations. For both flow-cases, the simulations using the Smagorinsky model are in good agreement with experiments, especially for the ten first modes. By contrast, the simulations with $k - \varepsilon$ show some strong differences with experiments from the fourth modes for the frictional case and the sixth mode for the non-frictional case.The value of the Elder coef ficient for the KE cases has little influence on the first and second modes, which is coherent with the length scale of the turbulent eddies associated to these modes (large 2D horizontal coherent structures). In contrast, the value of the Elder coefficient has strong influence on the next modes. For the frictional case, reducing the Elder coefficient leads to degrading the results, which indicates that the 3D turbulence is important even in the first modes. For the non-frictional case, the opposite behavior is observed, which is consistent with the physics. Indeed, in the non-frictional case, the flow is mainly driven by the horizontal geome try leading to large horizontal turbulent eddies. By increasing the Elder coefficient, the contribution of the

Figure 4. Sum of the mean fluctuating kinetic energy of the ten first modes, E_{10} , normalized for each flow case by their corresponding total kinetic energy, E_T . The black area corresponds to the contribution of the first mode; the white area corresponds to the cumulated contributions of $m = 6$ to $m = 10$; the grey areas correspond to $m = 2$ (dark

grey) to m = 5 (lighter grey). 3D turbulence is increased and prevents the 2D turbulence to plainly develop. It is interesting to notice that the previous behaviors observed from the third modes are slightly inversed for the two first modes: even though the modelled structures are 2D for the two first modes, a 3D contribution is required for approaching the experiments. This could represent an effect of the shallowness induced by the low water depth for both cases. The mean fluctuating kinetic energy distribution in the ten first modes is then compared to the total mean fluctuating kinetic energy of each flow-case in Figure 4. For the frictional case, the Smagorinsky model respects the best the energy distribution, while for the non-frictional case, the Smagorinsky model and the $k - \varepsilon$ models with $\lambda = 0$ and 0.08 gives relatively good results. For the frictional case, the 2D turbulence modelled by $k - \varepsilon$ is responsible for a too strong contribution relative to the 3D turbulence. 3.2 Temporal coefficients The temporal coefficients calculated with Equation 4 are represented in Figure 5 for experiments and numerical simulations. Notice that the first modes until at least m = 6 are representative of coherent structures (Peltier et al., 2014b) and they are therefore paired (Rempfer and Fasel, 1994). As a consequence, only the temporal coefficients for m = 1, 3 and 5 are represented here. For the frictional case, the amplitude and the frequency of the temporal coefficient of the first mode (and second, since they are paired) are the same for all simulations and they are very close to the experiment. From m = 3, differences appear with the experiment and between the turbulent closures. For both types of turbulent closures, the shape and the frequency of the

Figure 5. Temporal coefficients of the first, third and fifth

modes of the POD analysis.

temporal coefficients are not in good agreement with

the experiment. While with $k - \varepsilon$ models the oscilla

tions are regular during time and the amplitudes are

affected by the value of the Elder coefficient, the fre

quency and the amplitude of the temporal coefficient

may vary for a given mode with the Smagorinsky model. This could indicate that in the case of a fric

tional flow, turbulent closure as little influence on the

simulation result.Viscous effects could be also respon

sible for this behavior as the Reynolds number for this

flow case is relatively low (<10,000 see in Table 1).

The viscous effects prevent the complete development

of the turbulence, which is not taken into account in

the simulations.

For the non-frictional case, the Smagorinsky model

and the $k - \varepsilon$ model with $\lambda = 0$ (NF-KE0) give rel

atively good results for the first (and second) mode.

For the following modes, only the NF-KE0 case gives

good results. This result clearly confirms that the way

the 3D turbulent mixing is considered or not in the

modelling has tremendous impact on the simulation

results. Nevertheless it also confirms the dominance

of the 2D turbulence for non-frictional flows.

3.3 Spatial modes

The vorticity of the spatial modes calculated with

Equation 5 is used for describing the dynamics of Figure 6. Longitudinal profile of vorticity of the first, third and fifth spatial modes. the turbulent eddies represented by each pair of modes, whom temporal coefficients were presented in Figure 5. In Figure 6, the vorticity along the centerline of the reservoir (i.e. in the jet) is presented. For the frictional case, the first mode is relatively well represented with the Smagorinsky model and the $k - \varepsilon$ model, when the latter accounts for the 3D turbulence (F-KE and F-KE0.2). For the other modes, the $k - \varepsilon$ model generates structures smaller than in the experiment. In contrast, the Smagorinsky model enables a better description of the modes. For the non-frictional case, what was said about the turbulent closures for the temporal coefficients can also be applied for the spatial modes: suppressing or at least limiting the 3D turbulence enables to describe the six first spatial modes along the reservoir centerline. Nevertheless, just considering the vorticity at the centerline is not enough. As additional information, lateral profiles of vorticity are displayed for the same spatial modes in Figure 7. For both the frictional case and the non-frictional case, the simulations using the Smagorinsky model are in very good agreement with experiments. When using the $k - \varepsilon$ model, reducing the contribution of the 3D turbulence improves the modelling of non-frictional flows, while increasing the 3D turbulence improves the modelling of frictional flows.

Figure 7. Lateral profiles of vorticity of the first, third and fifth spatial modes.

4 CONCLUSION

The ability of the shallow water equations to model meandering flows in shallow rectangular reservoirs is investigated in this article.The choice of the best turbu lent closure is especially discussed. The first turbulent closure uses a $k - \varepsilon$ model for modelling the 2D hor izontal eddies, and an algebraic model for accounting for the 3D bottom-generated turbulence. The second turbulent closure is a subgrid-scale model account ing for the unresolved scales of the shallow water equations (Large Eddy Simulation-like model).

Two distinct flows in terms of friction regime were modelled (frictional and non-frictional). The frictional flow is mainly driven by the bottom-generated turbu lence (3D turbulence). While the non-frictional flow is driven by the reservoir geometry and is settled by large horizontal turbulent eddies.

The transient behavior of the meandering flow prevents the use of classical flow descriptors for comparing the simulations and the experiments. A Proper Orthogonal Decomposition of the fluctuating velocity fields was therefore used for describing the flow dynamics.

Comparisons between experiments and simulations emphasize the importance of modelling the small eddies for well representing meandering flows in shallow reservoirs.

Using the subgrid-scale model give the best results whatever is the friction regime of the flow. When using the $k - \varepsilon$ model coupled to the Elder model with different tuning parameter for the algebraic model, results emphasize the variable importance of the 3D turbulence in the flows. The contribution of the 3D turbulence must be decreased for non-frictional flows,

Dewals, B., Kantoush, S.A., Erpicum, S., Pirotton, M., Schleiss, A.J., 2008. Experimental and Numerical Analysis of Flow Instabilities in Rectangular Shallow Basins. Environmental Fluid Mechanics 8, 31–54.

Dufresne, M., Dewals, B., Erpicum, S., Archambeau, P., Pirotton, M., 2010a. Experimental Investigation of Flow Pattern and Sediment Deposition in Rectangular Shallow

Reservoirs. International Journal of Sediment Research 25, 258-270.

Dufresne, M., Dewals, B., Erpicum, S., Archambeau, P., Pirotton, M., 2010b. Classification of Flow Patterns in Rectangular Shallow Reservoirs. Journal of Hydraulic Research 48, 197-204.

Dufresne, M., Dewals, B., Erpicum, S., Archambeau, P., Pirotton, M., 2011. Numerical Investigation of Flow Patterns in Rectangular Shallow Reservoirs. Engineer ing Applications of Computational Fluid Mechanics 5, 247-258.

Dufresne, M., Dewals, B., Erpicum, S., Archambeau, P., Pirotton, M., 2012. Flow Patterns and Sediment Depo sition in Rectangular Shallow Reservoirs. Water and Environment Journal 26, 504-510.

Erpicum, S., Dewals, B., Archambeau, P., Pirotton, M., 2010. Dam Break Flow Computation Based on an Efficient Flux Vector Splitting. Journal of Computational and Applied Mathematics 234, 2143-2151.

Erpicum, S., Meile, T., Dewals, B., Pirotton, M., Schleiss, A.J., 2009. 2D Numerical Flow Modeling in a Macro Rough Channel. International Journal for Numerical Methods in Fluids 61, 1227-1246.

Hervouet, J.M., 2003. Hydrodynamique des écoulements à Surface Libre: modélisation numérique avec la méthode des éléments finis. Presse de l'Ecole Nationale des Ponts

et Chaussées, Paris, France.

Holmes, P., Lumley, J.L., Berkooz, G., Rowley, C.W., 2012.

Turbulence, Coherent Structures, Dynamical Systems and

Symmetry. Second Edition, Cambridge Monographs on

Mechanics. Cambridge University Press.

Kantoush, S.A., 2008. Experimental Study on the Influence

of the Geometry of Shallow Reservoirs on Flow Patterns

and Sedimentation by Suspended Sediments. Ph.D. the

sis, Ecole Polytechnique Fédérale de Lausanne (EPFL),

Lausanne, Switzerland.

Khan, S., Melville, B.W., Shamseldin,A.Y., Fischer, C., 2013.

Investigation of Flow Patterns in Storm Water Retention

Ponds Using Cfd. Journal of Environmental Engineering

139, 61–69.

Le Coz, J., Camenen, B., Peyrard, X., Dramais, G., 2012.

Uncertainty in Open-Channel Discharges Measured with

theVelocity-Area Method. Flow Measurement and Instru

mentation 26, 18–29. Peltier,Y., Erpicum, S.,Archambeau, P., Pirotton, M., Dewals, B., 2013. Experimental and Numerical Investigation of Meandering Jets in Shallow Reservoir: Potential Impacts on Deposit Patterns, in: Two-pHase modElling Sediment dynamIcs in Geophysical Flows, THESIS 2013, June 10- 12, Chatou, France. Peltier,Y., Erpicum, S.,Archambeau, P., Pirotton, M., Dewals, B., 2014a. Experimental Investigation of Meandering Jets in Shallow Reservoir. Environmental Fluid Mechanics 14, 699–710. Peltier,Y., Erpicum, S.,Archambeau, P., Pirotton, M., Dewals, B., 2014b. Meandering Jets in Shallow Rectangular Reservoirs: Pod Analysis and Identification of

Coherent Structures. Experiments in Fluids 55, 1740. Peltier,Y., Erpicum, S.,Archambeau, P., Pirotton, M., Dewals, B., 2014c. Meandering Jets in Shallow Rectangular Reservoir, in: 7th International Conference on River Hydraulics, River Flow 2014, September 2–5, Lausanne, Switzerland. Peltier,Y., Erpicum, S.,Archambeau, P., Pirotton, M., Dewals, B., 2015. Can Meandering Flows in Shallow Rectangular Reservoir Be Modelled with the 2D Shallow Water Equations? Journal of Hydraulic Engineering 141, 04015008. Peng, Y., Zhou, J.G., Burrows, R., 2011. Modeling FreeSurface Flow in Rectangular Shallow Basins by Using Lattice Boltzmann Method. Journal of Hydraulic Engineering 137, 1680–1685. Perrin, R., Braza, M., Cid, E., Cazin, S., Barthet, A., Sevrain, A., Mockett, C., Thiele, F., 2007. Obtaining Phase Averaged Turbulence Properties in the near Wake of a Circular Cylinder at High Reynolds Number Using Pod. Experiments in Fluids 43, 341–355. Rempfer, D., Fasel, H.F., 1994. Evolution of ThreeDimensional Coherent Structures in a Flat-Plate Boundary Layer. Journal of Fluid Mechanics 260, 351–375. Roache, P.J., 1997. Quantification of Uncertainty in Computational Fluid Dynamics. Rodi, W., 1980. Turbulence Models and Their Application in Hydraulics:A State of the Art Review. IAHR book publications, Delft, the Netherlands. Stovin, V.R., Saul, A.J., 2000. Computational Fluid Dynamics and the Design of Sewage Storage Chambers. Journal of the Chartered Institution of Water and Environmental Management 14, 103–110. Wark, J.B., Samuels, P.G., Ervine, D.A., 1990. A Practical Method of Estimating Velocity and Discharge in Compounds Channels, in: International Conference on River Flood Hydraulics, September 17–20 Sept, Wallingford, England. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Turbulent momentum exchange over a natural gravel bed

C. Miozzo & A. Marion

Dipartimento di Ingegneria Civile, Edile ed Ambientale, Università degli Studi di Padova, Padova, Italy

A. Nichols & S.J. Tait

Department of Structural Engineering, University of Sheffield, Sheffield, UK

ABSTRACT: Exchange processes occurring between a porous bed and the stream flow above play an important

role in controlling the transport of contaminants and other substances in rivers and streams. This work examines the vertical momentum transfer occurring above the roughness layer of a uniform gravel bed for three shallow uniform turbulent flows. PIV measurements were used to calculate the boundary shear stress via the double average method. The data was also used to define the different roles of the bed shape and of the flow conditions in controlling the momentum flux. Employing a cross-correlation technique it was observed that the bed shape strictly controls the spatial pattern of the momentum flux in the zone above the bed in all three cases, and that the flow conditions are less influential over the near-bed momentum flux pattern for the analysed flow conditions. These initial results may be generalised to other flows, therefore it is hoped that this study will stimulate further investigation on the subject.

1 INTRODUCTION

Turbulent flow over porous media is a topic with relevance to a number of environmental processes in rivers and streams. In particular, transfer phenomena at the fluid/porous interface is an important topic of research because of its wide range of applications in estimating the fluxes of dissolved organic molecules, inorganic ions, and gases between a porous bed surface and the overlying fluid. This is important because permeable streambeds can act as sinks for harmful toxicants and fine sediments, and can also influence oxygen transport (Zhou and Mendoza, 1993).

It has been theorised that the migration of solutes between streams and flat gravel beds is governed by the turbulent momentum transfer across the stream–subsurface interface (Packman et al., 2010; Zhou and Mendoza, 1993). Boundary shear stress is a measure of the average momentum removed from a moving fluid by a boundary, and it dictates the momentum available from the fluid to entrain, transport and deposit material at the fluid-bed interface. This paper aims to charac terise the spatial pattern of the momentum exchange between a porous rough boundary and the flow in the zone just above.

As the relative contribution of factors controlling the spatial pattern of the momentum flux are still unknown, it was hypothesised that it is predominantly a function of the bed profile and the flow conditions. This paper presents experiments carried out in a lab oratory flume to determine any correlation between these factors and the vertical momentum flux pattern. 2 THEORETICAL BACKGROUND 2.1 Physical components of boundary shear stress In order to gain a better physical insight into the mechanisms that contribute to the overall transfer of momentum at a given vertical location in the flow, the velocity data was examined by decomposing the timeaveraged variables into spatially-averaged (denoted by angle brackets) and spatially fluctuating (denoted by a wavy overbar) components, such that $\bar{u}_i = \langle \bar{u}_i \rangle + \bar{u}_i$, where u_i is the instantaneous velocity in the i-th direction. The spatial fluctuations in velocity hence result from the difference between the double-averaged $\langle \bar{u}$

$i\rangle$ and the time-averaged $\bar{u}_i$ values, $\bar{u}_i = \bar{u}_i - \langle\bar{u}_i\rangle$, similar to the conventional Reynolds decomposition. By averaging over a spatial domain, the decomposition can be used to estimate all the forces on a rough, sediment boundary under an averaging area A_0. For a 2D, steady, uniform flow over a static boundary with a variable vertical porosity, the total force F_b on the boundary under area A_0 can be defined as: where:

F_{fd} and F_{vd} are the form drag and viscous drag, p is fluid pressure and $n = n_x, n_y, n_z$ is the unit vector normal to the surface S of the section of bed described by the area A_0.

An important parameter in Equation 1 is the roughness geometry function ϕ, which is defined as A_f / A_0, where A_f is the area of water in the averaging domain at y and A_o is the area at y_c. This function is a measure of the variation in the geometrical properties of the boundary in all three dimensions and it enables the influence of the surface geometry on the momentum equation to be extended below the crests of the roughness elements and down to the roughness trough. This is not possible with the Reynolds equations. The dependency of F_b (and therefore boundary shear stress) on ϕ in Equation 1 proves that the double averaging approach provides a consistent way to link the spatially-averaged boundary shear stress to the vertical variations in the bed surface geometry.

For the turbulent flow field above the roughness layer, where $-\langle\tilde{u}\tilde{v}\rangle = 0$, $F_{fd} = 0$ and $F_{vd} = 0$ and $\phi = 1$, Equation 1 is equivalent to that derived from the Reynolds equations.

2.2 Contributions to the total fluid force

Equation 1 can be used to understand the relative importance of the different momentum transfer mech anisms. Over a defined plan area, A_0, between y_w and y_c, the Reynolds stress in the fluid almost completely balances the total force supplied by the fluid, i.e. the force that is transferred from the flow above the bed into the fluid and bed elements within the roughness layer.

This phenomenon is due to turbulent fluid exchange. At a very small distance above the rough ness crest, the Reynolds stress begins to reduce. Here, a small additional force caused by form-induced stress is transferred towards the roughness trough because of the presence of persistent vortices behind roughness elements.

This also represents the upper effective boundary of the roughness layer, where the flow is signifi cantly spatially heterogeneous. In this zone, force is transferred from the flow above by both turbu lence and spatial heterogeneity in the time-averaged flow. At the upper height of the roughness layer, the total fluid force is a reflection of the total momen tum exchange between the roughness layer and the

overlying flow. Within the roughness elements, at a

level y, there is an additional force of $\rho g S_b \phi A_0 (y_c - y)$

caused by the weight of the fluid in the roughness

layer, and this causes the total force to increase.

Surface form drag and fluid forces caused by tem

poral and spatial velocity variation balance this addi

tional force. This introduces a major change in the

mechanisms of momentum exchange. Total force at Figure 1. Flume overview. y, $\rho g S_b A_0 [(y_w - y_c) + \phi(y_c - y)]$, is thus partly transferred further down by turbulent exchange and spatial heterogeneity in the time-averaged flow, and partly extracted by form drag with the bed surface. Within this region, the vertical variation of the force contributed by Reynolds and form-induced stress and form drag closely resembles the change in the roughness function ϕ and is therefore expected to be controlled by the geometry of the deposit. 3 EXPERIMENTAL FACILITIES AND FLOW CONDITIONS A series of experiments was conducted in which a range of steady, uniform shallow flows was established over a rough boundary in order to measure the nearbed momentum flux. This section will describe the experimental setup, instrumentation and measurement techniques, and the range of flow conditions recorded. 3.1 Flume setup 3.1.1 Flow control and bulk measurements The experiments were carried out in a 12.6 m long, sloping rectangular flume which was 459 mm wide. The gradient of the flume was fixed at 0.002. The experimental setup is represented in Figure 1. The magnitude of the discharge was determined using a u-tube manometer connected to a standard orifice plate assembly (BS5167-1, 1997). An adjustable gate was placed at the downstream end of the flume to control the uniform flow depth, which was measured with point gauges at either end of the flume 3.1.2 Bed type and bed measurement The flume contained a bed of well-mixed and washed river gravel, which was scraped to a uniform thickness of d_g = 50 mm (nominal) so there were no significant topographical features or bedforms. The gravel particles had a density of ρ_g = 2600 kg/m^3 and mean grain size (by mass) of d_{50} = 4.4 mm. The grain size was approximately normally distributed (Nichols, 2013). To compare the momentum data with the bed profile, the bed shape was measured via a laser displacement sensor (LDS), Before and after the tests, the LDS was used to measure the bed surface elevation. The profiler was a Keyence LK-G82 laser displacement sensor, which is stated to be accurate to within ±0.25 μm, with a spot diameter of 45 μm. Bed elevation was recorded at a spatial resolution of 0.5 × 0.5 mm.

Figure 2. Diagram of camera arrangements for flow visualisation.

To remove erroneous high or low readings in the LDS data, a two-dimensional median filter was used, which was 3 × 3 in size (1.5 mm × 1.5 mm). This area is smaller than the grain size in order to avoid smoothing or removal of real bed features. The bed structure correlation and the probability density func tion were thereby calculated. The bed data show a bell-shaped distribution about the mean elevation, with a maximum elevation of around 4 mm from the mean (Nichols, 2013). The bed scans also showed that there was no significant grain motion during the tests.

3.2 Particle Image Velocimetry (PIV)

The data used to quantify the velocity field were recorded with a Dantec Dynamics two-dimensional Particle Image Velocimetry (PIV) system, which uses two pulsed Nd:YAG lasers to illuminate and visualise particle motion in a plane within the flow (Nichols, 2013).

3.2.1 System setup

Figure 2 presents a diagram of the camera arrangement for flow visualisation. A laser light sheet illuminated a volume approximately 220 mm long in the stream wise direction and approximately 3 mm thick in the lateral direction. Two CCD cameras, each with an image area of 1600 × 600 pixels, were focused on the laser sheet and synchronised with the laser pulses. The overlapping field of view of the two cameras cov ered an area in the laser plane of 247 × 89 mm. The resulting resolution of the images was approximately 42 pixels/mm ^{2} .

The seeding particles introduced into the flow to perform the PIV measurements were Plascoat Talis man 30, which have a diameter of around 150 μm (Hunter, 2010) and a narrow particle size distribution (Plascoat, 2013). Being almost neutrally buoyant, with a specific gravity of 0.99, these particles maintain sus pension for several hours (Vlaskamp, 2011) following the flow path representatively.

In order to reduce any light pollution in the images, the cameras were equipped with narrow band-pass filters which passed light at 532 ± 2 nm; the light obtained was then the light reflected by the particles only.

Each camera captured a pair of particle images sep

arated by a time delay of 1 ms at a fixed frequency of

26.9 Hz. For each measurement, images were captured for a duration of 5 minutes, in order to generate a time series of image pairs on each camera. 3.2.2 Calibration The measured velocities were in pixels/s at spatial locations defined in pixels, and it is more practical for further calculation to transform them into m/s and m respectively. Therefore, a calibration was performed to allow the output of the PIV analysis to be represented in real terms. The calibration procedure involved the capturing of images of a 200 × 200 mm 2 calibration plate, which consisted of an orthogonal grid of circular markers at known spatial positions. For calibration, the plate was placed in the plane of the laser, at the centre of the camera's field of view, immersed in water so that any refraction effects were captured as they would be for flow conditions. Several images were captured on each camera, and an image-model fit was then performed using a direct linear transform to determine the calibration constants, which are to be applied to the raw PIV data. This type of transform is suitable for applications such as this where any refractive boundary (the glass wall of the flume) is planar. 3.2.3 Measuring the datum position relative to the bed It was important to ensure that the spatial frame of reference was the same for the PIV data and the bed elevation scan. It was therefore necessary to determine the streamwise position of the PIV datum relative to the bed elevation measurements, and also the vertical distance between the mean bed position and the PIV datum. The laser displacement sensor was used to scan the bed surface in the plane of the PIV laser, in order to calculate the mean bed position relative to the LDS sensor. A multi-level target was then measured by both the LDS sensor and the PIV cameras in order to calculate the offset between the PIV system and the LDS system. This procedure is described in more detail by Nichols (2013). This also enabled the PIV data below the bed profile to be masked using the scanned bed profile. 3.2.4 PIV data processing Each image pair captured by the two PIV cameras was divided into interrogation areas of 32 × 32 pixels (with 50% overlap). This interrogation area size corresponds to a physical area of around 4.9 × 4.9 mm, with the overlap meaning the spatial resolution of the measurements is around 2.5 mm in both the streamwise and vertical directions. For each interrogation area the mean flow vector was calculated, resulting in a vector field of dimensions 92 × 34 vectors (247 × 89 mm 2). The vector maps then underwent range validation and moving average validation in order to correct any spurious data points (Nichols, 2013). Finally, the vector maps from the two PIV cameras were combined to form the final vector field. In this way a time series of vector maps was constructed for each of the flow conditions described in Table 1. This data was

Table 1. Measured hydraulic conditions for gravel bed flows. The table shows also the maximum magnitude correlation coefficients calculated for each flow condition. The functions are represented in Figure 5.The correlation function is non-dimensional.

Condition	Bed Slope S_0 [-]	Depth D [mm]	Velocity V [m/s]	Equivalent Roughness k_s [mm]	Relative Submergence D/k_s [-]	Correlation coefficient
1	0.002	60	0.32	9.1	6.6	0.30
2	0.002	70	0.35	9.1	7.7	0.35
3	0.002	80	0.40	7.4	10.8	0.25

then exported in a numerical format to allow detailed analysis using Matlab.

3.3 Experimental conditions

A range of three flow depths from D = 0.06 m to 0.08 m was analysed (seeTable 1).This range was chosen since it represents a subset of typical submergences found in gravel bed rivers (Ferguson, 2007; Robert, 1990), and in order to avoid strong lateral components which can become significant when the depth exceeds around 1/5 of the channel width (Nakagawa & Nezu, 1993). A slope of $S_0 = 0.002$ was set as it represents a typical bed slope found in gentle gradient streams (Rosgen, 1994).

Since the depth is measured to the nearest 0.5 mm and the flow rate to the nearest 0.5 l/s, it can be shown that the calculation of depth-averaged velocity, V, is hence accurate to the nearest 0.01 m/s.

The equivalent roughness height, k_s, for these conditions was determined using the Colebrook-White equation modified for open channel flows (Barr, 1963). This formula was used since it conveys the physical reality that deeper flows experience a lower resistance, and it is therefore more sensitive to the flow conditions (by definition it characterises the resistance to flow) than Manning's equation which gives a more general roughness coefficient for a given physical channel. The Colebrook-White equation for open channels is presented as:

where:

is the Darcy-Weisbach friction factor, S_0 is the energy

slope, g is the acceleration due to gravity and Re is the

Figure 3. Streamwise and vertical mean velocity profiles (u, v) for flow conditions 1, 2 and 3: $S_0 = 0.002$; D = 60, 70, 80 mm respectively. V = 0:32; 0:35; 0:40 m/s respectively.

depth-based Reynolds number. This number is calculated from the measurement of discharge and mean water depth, which are each accurate to 0.5 l/s and 0.5 mm, respectively. The Reynolds number is therefore computed to two significant figures as shown in the tables. The range of flow depths was simulated so that the ratio of depth to equivalent roughness height (D/k_s) varied from 6.6 to 10.8, which is within the range of relative submergence values found in gravel bed rivers without appreciable bed forms by Ferguson (2007). The shear velocity is used later for nondimensionalising, but it is not shown in Table 1 for brevity. It can be easily calculated from the depth and bed slope data presented in 1994). as: 4 ANALYSIS 4.1 PIV data validation In order to assess the accuracy of the processed data, the time-and-space-averaged velocity profiles and the turbulence intensity profiles were calculated in the streamwise and vertical directions. For all three flow conditions the mean velocities are plotted in Figure 3 in order to visualise the shape of the profiles. To evaluate the error in these values, the profiles were first calculated for every vertical column of PIV interrogation areas (i.e. every spatial location in the streamwise direction). These were then averaged to determine the double (space and time) average. The standard deviation of the time average at each depthwise position was calculated to give an indication of variability across the measurement frame. This variability was never systematic (mean velocity increasing in upstream or downstream direction for example), but was more random, indicating that this either represents normal measurement error or the true spatial variation. These variations are represented by the error bars in Figure 3.This figure shows the vertical mean velocity being close to zero, and the streamwise velocity increasing approaching the free surface. The streamwise velocity also agrees with the mean velocity

Figure 4. Streamwise velocity defect form for flow condi

tions 1, 2 and 3: $S_0 = 0.002$; D = 60, 70, 80 mm respectively.

measurements of Table 1. For all the flow conditions,

the defect velocity $(U_{max} - U) / U_*$ is plotted against

the normalised depthwise location, y/D, in Figure 4, in

order to compare it with the expected profile described

by Nakagawa & Nezu (1993):

where U is the time-space averaged velocity in the streamwise direction, U_{max} is the maximum of the time-space averaged velocity in the same direction, $U*$ the shear velocity (see Eq. 6), and $k = 0.41$ is the universal Von Karman constant. The expected profiles match well with the experimental data.

4.2 Boundary shear stress computation

At the upper height of the roughness layer, the total fluid force is a reflection of the total momentum exchange between the roughness layer and the overly ing flow (Cooper & Tait, 2010). Following the results by Cooper and Tait (2010), the total force F_b (Eq. 1) under A_0 (the averaging area) is divided by A_0 and evaluated right above the roughness tops, where the roughness geometry function is $\phi = 1$:

Similarly, the values of the boundary shear stress per unit area τ were calculated using the measurements of the velocity in the streamwise and vertical direc tions. The timeand space-fluctuations of the velocity were computed using the Reynolds decomposition $u = \bar{u} + u'$ for instantaneous variables and the decomposition $\bar{u} = \langle \bar{u} \rangle + \tilde{u}$ for time-averaged vari ables. These values were then multiplied elementwise in order to obtain a time series of the desired physical characteristic: $\tau = -\rho[u' v']$ and $\bar{\tau} = -\rho[\bar{u}' \bar{v}' + \tilde{u}' \tilde{v}']$.

No viscous term appears in Equation 9 because in high

Reynolds number water flows this stress is orders of

magnitude less than the turbulent stress. Figure 5. Contour plot of the time-averaged time series of τ, which corresponds to the momentum flux per unit time, for the gravel bed conditions. The darkest area is the mask applied to hide the data below the bed. The mean (in space and time) boundary shear stress was hence calculated for all the flow conditions. The results confirmed that the shear stress above the roughness crest is almost entirely due to Reynolds' stresses. Afterwards, the time series of τ was averaged in time, obtaining the momentum flux per unit time at each spatial point in the flow field. 5 RESULTS Contour plots of the momentum flux are plotted in Figure 5. The plots show the lower 25% of the lowest flow depth over the bed (15 mm), in order to understand the importance of the studied scale in the area of interest. Comparing the three images, there is a great deal of similarity, suggesting that the flow next to the bed seems to be affected by the bed itself more than the flow conditions. Indeed, careful observation of the momentum flux intensity shows thata strong correlation exists among the different depths. 5.1 Cross-correlation A cross-correlation technique was used in order to determine whether there is a spatial correlation between the momentum flux and the bed shape. In general, given two random vectors, a(n) and b(n), the Matlab function xcorr estimates the cross correlation sequence of a random process, i.e. returning a vector with the estimated covariance $\hat{R}_{ab}$ at a certain position m and a vector of the lag-indices at which the covariance was estimated. It is assumed for discussion that a n and

Figure 6. Detrended momentum flux and bed shape vector,

and results of their cross correlation for condition 1, 2 and 3.

Correlation coefficients are non-dimensional.

b n are indexed from 0 to (N − 1), and $\hat{R}_{ab}(m)$ from −

(N − 1) to (N − 1). This gives a measure of the similar

ity of two signals as a function of a spaceor time-lag

applied to one of them. As, in general, the correlation

function requires normalisation to produce an accu rate estimate, the function was set in order to return

the estimated covariance normalised by the product of

the standard deviations of the two vectors:

and similarly for $m < 0$.

The row of the $\bar{\tau}$ matrix above the roughness tops

(located at the height of 4.7 mm) and the bed shape vec

tor were cross-correlated to determine the maximum

(or minimum, whichever has the largest magnitude)

amplitude of the spatial cross-correlation function.

The vectors are represented along with the result of

the cross correlation in Figure 6 (note that the bed

shapes are not to scale, in order to more easily allow

comparison with the momentum flux profile). Figure 7. Cross correlation function for the gravel bed conditions. Correlation coefficients are non-dimensional. Figure 8. Spectral analysis for the three flow conditions. Both the momentum flux and the bed profile were detrended (linear trend removed) before being crosscorrelated. The cross correlation functions are plotted in Figure 7. The momentum fluxes shown in Figure 6 exhibit a similar pattern independently from the depth in the three flow conditions. Therefore, a strong correlation between momentum flux and bed shape was expected. The largest correlation coefficients are reported in Table 1 for each flow condition. These correlation values are not negligible, suggesting there is a link, but they are relatively low, suggesting that the link is more complex than a direct linear relationship between bed shape and momentum flux. 5.2 Spectral analysis In Section 5.1, the cross correlation was calculated to identify whether the momentum flux profile relates to the bed profile. Afterwards the power spectra of the cross correlation functions were computed to examine the scales at which the bed and the momentum flux interact. The Fast Fourier Transform (FFT) of the correlation coefficients would show any strong periodicity in the correlations, which suggests a shared periodicity in the two variables. In particular it was expected that there might be grain-scale correlation and depth-scale correlation. The absolute value of the FFT computed for the cross correlation functions is plotted in Figure 8 for Conditions 1, 2 and 3. The values of the largest peaks are reported in Table 2 along with their spatial period.

Table 2. Amplitude (in absolute value) of the largest peaks calculated for the three flow conditions (Figure 8) and for the gravel bed profile spectra (Figure 9), and corresponding spatial periods. The values of the spatial periods are the same.

Condition 1				
Amplitude	0.0758	0.0421	0.0319	0.0257
Spatial Period	0.0316	0.0430	0.0175	0.1184
Condition 2				
Amplitude	0.0737	0.0424	0.0409	0.0268
Spatial Period	0.0316	0.0430	0.0215	0.1184
Condition 3				
Amplitude	0.0560	0.0315	0.0305	0.0113
Spatial Period	0.0316	0.0430	0.0215	0.1184
Bed spectra				
Amplitude	0.0888	0.0618	0.0609	0.0607
Spatial Period	0.0316	0.0430	0.0175	0.0215

Figure 9. Spectral analysis of the gravel bed profile.

The spectra show several strong length scales, which indicates that the momentum field is somewhat complex. Even in this case though, the spatial period represented by the strongest peak is the same for the three depths, 32 mm. This suggests that the depth does not affect the spectra in a consistent way.

Although the momentum transfer pattern is apparently complex for the gravel bed, the cross correlation functions shown in Figure 8 look similar for the three depths, suggesting again that the bed shape is controlling the momentum flux rather than the flow Reynolds number.

These functions are in some way periodic, even if there is more than one dominant period, as confirmed by the resulting spectra. If the bed was truly randomly organised then the only observed periodicity should occur at the grain scale. In order to understand whether there is a periodicity characterising the bed profile, its auto-correlation function was computed and then the spectrum of the auto-correlation coefficients was calculated (see Figure 9).

The absolute value of the first four peaks is reported in Table 2 along with their spatial periods. It is interesting to note that the spatial periods correspond to the ones calculated for the cross-correlation function between the gravel bed and the momentum flux.

Hence, it seems that the bed contains some spatial coherence beyond the grain scale, and thereby strongly controls the momentum flux with the overlying flow, independently from the flow conditions. To explain the maximum dominant scale at 44 mm, a hypothesis could be the interaction of the subsurface flows within the bed structure (which was approximately 50 mm thick) with the upward and downward flux of fluid. If this is the case, subsurface flows may behave in a similar way to secondary flows – forming cells with the same scale both horizontally and vertically. It could be possible then, that the horizontal scale is limited by the vertical scale which itself is limited by the thickness of the bed layer (d_g = 50 mm). 6 CONCLUSIONS A series of experiments was carried out in a tilting rectangular laboratory flume in order to investigate the momentum transfer between a turbulent flow and a porous bed. In particular, the goal of this study was to quantify and compare the spatial pattern of the momentum flux with the spatial pattern of the bed, and to calculate the scale of the phenomenon. Three flow conditions were selected for this study. It is believed that for turbulent shallow flows the momentum transfer occurring between the bed and the overlying flow is predominantly controlled by the bed shape and the bed particles. This is found to be true for the case of a gravel bed, characterised by a random distribution of particles. It is clear that additional work will be required before a complete understanding of this phenomenon occurs, especially providing more detailed measurements in the zone examined in this paper. The experimental setup described in Section 3 could be set to get a better insight on the 20% of the flow depth right above the bed. Then it might be possible to examine the role of the bed structure in threedimensional flow characteristics, in order to confirm the results described so far. The cross-correlation functions between the nearbed momentum flux and the physical bed profile were characterised by the same periods independently from the flow depth, suggesting that the flow conditions did not significantly affect the organisation of the momentum flux. The disorganised gravel bed shape contains more than one dominant length-scale, but there is not a dominant spatial period coincident with the diameter of the bed particles (d_{50}). It is of note that the spectra of the momentum flux have the same periodicity that is presented by the bed structure. Therefore, it is suggested that the shape of the bed and the individual particles control the momentum transfer between the bed and the overlying fluid rather than the flow conditions. It was also found that the dominant length scale in the momentum flux may be governed by flow cells within the gravel bed, and thereby relate to the bed thickness. This is true for the range of flow conditions examined in this study. In conclusion it is hoped that this work will stimulate further investigation in this field.

Barr, D.I.H. 1963. Tables for the Hydraulic Design of

Pipes, Sewers and Channels. Hydraulic Research Station, Wallingford, UK.

Cooper, J.R. & Tait, S.J. 2010.Examining the physical com ponents of boundary shear stress for water-worked gravel deposits, Earth Surface Processes and Landforms, 35(10), 1240–1246.

Ferguson, R. 2007. Flow resistance equations for gravel and boulder-bed streams. Water Resources Research, 43(5).

Hunter, D.J. 2010.Volumetric Flow Prediction using Multiple Plane Particle Image Velocimetry. PhD Thesis, School of Engineering, University of Warwick, Coventry, UK.

Nakagawa, H. & Nezu, I. 1993. Turbulence in Open Channel Flows, Taylor & Francis.

Nichols, A. 2013. The interaction mechanism of airborne acoustic fields with turbulent shallow flows, School of Engineering, Design and Technology, PhD Thesis, University of Bradford, Bradford. Nikora, V., McEwan, I., McLean, S., Coleman, S., Pokrajac, D. & Walters, R. 2007. Double-Averaging Concept for Rough-Bed Open-Channel and Overland Flows: Theoretical Background. Journal of Hydraulic Engineering, ASCE, 133(8), pp. 873–883. Packman, A., Salehin, M., and Zaramella, M., 2004. Hyporheic Exchange with Gravel Beds: Basic Hydrodynamic Interactions and Bedform-Induced Advective Flows. J. Hydraul. Eng., ASCE, 130(7), 647–656. Robert, A. 1990. Boundary roughness in coarse-grained channels. Progress in Physical Geography, 40, pp.42–70. Rosgen, D.L. 1994. A classification of natural rivers, Catena, 22, pp.169–199. Vlaskamp, J.H.A. 2011. An experimental and numerical investigation into the vertical dependence of Taylorcolumns generated by a rotating disc, PhD Thesis, School of Engineering, University of Warwick, Coventry, UK. Zhou, D. and Mendoza, C., 1993. Flow through Porous Bed of Turbulent Stream. J. Eng. Mech., ASCE, 119(2), 365-383.

Effects of changes in flow velocity on the phytobenthic biofilm below a small scale low head hydropower scheme

L.O'Keefe & S. Ilic

Lancaster Environment Centre, Lancaster University, Lancaster, UK

ABSTRACT: This study presents a spatial analysis of physical and biotic river conditions below a low head hydro scheme in the River Goyt, UK. The overall aim was to assess whether changes in localized hydrological features, introduced by low head hydro, affect phytobenthic biomass. Single point and profile measurements of flow velocity and velocity vectors, elevation, depth and biofilm biomass were mapped. Results showed evidence of high flow velocity on the hydro side of the river and low flow velocity on the non-hydro side of the river. A number of distinct hydrological and morphological features were defined. Biofilm biomass appeared lower on the hydro side of the channel but no obvious relationship with flow velocity was observed. Future analysis will include evaluation of phytobenthic species on either side of the river channel and an investigation into the combined effects of a number of variables on phytobenthic biomass.

1 INTRODUCTION

In recent years (2009–2015) there has been renewed interest in hydropower in the UK (Demars and Britton, 2011; Robson et al., 2011, Anderson et al., 2014). This directly corresponds to renewable energy targets

(Fraser et al., 2015), technology advances (Bracken and Lucas, 2011) and financial incentives (Fraser et al., 2015). Namely the UK's target to produce 15% of its energy from renewables by 2020, new energy efficient turbines like the Archimedes Screw Turbine and the Feed in Tariff which provides payments to owners of renewable schemes based on the amount of electricity sold to the national grid (Fraser et al., 2015). Current trends in hydropower applications suggest that the most popular schemes in England are low head designs where the height difference between the intake and outlet is less than 5 meters and where the turbine is situated directly on top of or directly adjacent to an existing weir (Fraser et al., 2015) 2015). The main focus of this paper is on low head hydro and from this point on the different layouts will be referred to as 'on weir' and 'by weir' schemes respectively. Figure 1 displays a schematic diagram of the two different lay outs for 'on weir' hydro where the turbine is situated directly on top of a weir and 'by weir'where the turbine is situated directly adjacent to a weir. Unfortunately understandings of the aquatic impli

cations of low head hydro has not kept pace with the recent rise in the number of development proposals and even though it has been suggested that low head hydro is "environmentally benign" (Paish, 2002) there is a huge lack of evidence available to support this claim Figure 1. Schematic diagram of low head hydropower designs for ‘on weir’ and ‘by weir’ hydro respectively (adapted from EA, 2013). (Robson et al., 2011 and Anderson et al., 2014). There is an urgent need to conduct detailed investigations to develop current understandings (Robson et al., 2011). To date review studies have derived the potential implications from the relatively well known impacts of high head schemes. Conclusions are often based on expert opinion rather than experimental data and the impact of the weir alone is typically separated from the additional impacts of the scheme (Anderson et al., 2014).As schemes are often installed on existing weirs understanding the impacts of the weir on natural

river conditions is crucial. The hydrological, morpho logical and biotic impacts of weirs are relatively well researched but the additional impacts or added benefits of the scheme are still uncertain.

Weir structures by their very nature can change nat ural river conditions. They can create a weir pond directly above the weir and a scour pool directly below the weir. This paper is concerned with the area directly below the weir where the cascading water erodes the river bed and banks. This area typically consists of a deep, wide pool with high energy, complex flow and a mid-channel island or tail riffle were the bed material is deposited as energy in the flow decreases (Mould et al., 2015). This area is often associated with specialized aquatic communities adapt to the complex environ ments. In line with the Environment Agency (Mould et al., 2015) this whole area will be referred to as the weir pool from herein.

Adding a scheme to a weir could cause changes in the weir pool environment. Robson et al., (2011) suggests that by changing the distribution of flow a scheme might cause changes in energy dissipation, flow pattern, morphology and aquatic communities.
A modelling study conducted by the Environment Agency (Mould et al., 2015) supports this theory suggesting that there could be changes in the spatial variation of flow velocities and depths. Although they do conclude that such changes are unlikely to alter habitats.
A case study at Romney Weir on the River Thames revealed higher flow velocities along the river bank closest to the turbine and lower velocities on the opposite side of the river up to 20 meters beyond the structure (Mould et al., 2015). In this particular instance the Environment Agency (Mould et al., 2015) concluded that the changes in flow were not “ecologically significant” but such conclusions are based on expert opinion and species preference rather than observed evidence. In-situ biotic investigations need to be carried out before such claims can be accepted. Different communities could potentially develop either side of the river, especially sessile benthic communities, like the phytobenthos which are unable

to move. Particular changes in communities might occur were the flow from the main river channel and turbine outlet collides. There could be changes in hydrological and morphological features which in turn could alter biotic communities. Similarities can be drawn to the interface of two flows at river confluences and trib utaries. However this is most likely to occur at 'by weir' schemes where the flow is diverted through a turbine forebay, a small channel in which the turbine is situated. Where the water is discharged back into the main river channel and the two flows collide any of the following features could occur;

1. Stagnation at the upstream junction corner between the outlet and main channel,

2. Mixing, development of shear layers and scouring of benthic communities where the two flows combine, accelerate and scour the river bed, 3. Separated flow below the downstream outlet channel junction corner and bar formation, 4. Deflection where the flows collide and change path, 5. Advanced recovery downstream (adapted from Szupiany et al., 2009). Further morphological changes could occur were the sediment that would have built up behind the weir will pass through the turbine forebay. This suggested by Anderson et al., (2014) and has been named the draw down effect. This study will explore the potential aquatic implications of a low head 'by weir' schemes in a bid to improve knowledge and understandings, update and improve empirical evidence and to inform scheme designs to reduce their impact on the environment. The main motivation is to identify distinct hydrological, morphological and biotic features which could be attributed to the scheme. 2 METHODS A two stage field campaign was designed in order to understand the hydrological, morphological and biotic features below a low head 'by weir' scheme. The first stage involved a spatial survey aimed at mapping single point measurements of flow velocity, bed elevation and

phytobenthic biomass. The phytobenthic community was chosen as a study species as it has many attributes which make it well suited to biomonitoring (Law, 2011). The phytobenthic community is easy to measure, collect, handle and store. Being sessile it is likely to change in response to changes created by the hydro scheme. Sitting at the base of the food web changes in its biomass will have far reaching effects on the rest of the food chain. High flow velocity and shear stress are often associated with scouring of the biofilm and reduced biomass. High flow velocity can roll cobbles and boulders and cause the phytobenthos to become detached (Law, 2011). The second stage involved stationary Acoustic Doppler Current Profiler (ADCP) measurements and species analysis of samples collected from the near and far side hydro river banks. This paper presents results from the single point measurements and an initial analysis of the ADCP measurements. 2.1 Study site The field campaign was conducted in the River Goyt, UK, below Stockport Hydro (SJ936789441). Stockport Hydro was installed in 2011 and became operational in October 2012. The scheme is a low head ‘by weir’ scheme and consists of twin Archimedes Screw Turbines and a fish pass. The scheme has a “Hands Off Level” (HOF) of 6cm which means that the scheme can divert 100% of the flow as long as 6cm is maintained on the weir crest. Stockport Hydro monitors the abstraction rate of the scheme and records the level on the weir every 15 minutes. This data was utilized during

Figure 2. Survey design for assessing the aquatic implica

tions of a low head ‘by weir’hydro scheme on the River Goyt,

UK (SJ936789441) were dashed lines represent concrete

structures, crosses display the locations of measurements and

diagrams a) and b) represent the spot point measurements and

flow profile measurements respectively.

the field campaign. It must be noted that only one

turbine was operational during data collection. This

turbine was closest to fish pass on the hydro-side river

bank. The site has a well-defined island in the middle

of the river covered in vegetation and a number of permeable concrete structures on the far side river bank (non-hydro side).The concrete structures are rep resented by dashed lines in Figure 2. The concrete structures were not installed as part of the scheme. Figure 2 shows the outline of the area surveyed. The turbines sit in the small channel adjacent to the main channel known as the turbine forebay. The outlet is the point where this channel meets the main river channel and were the water is discharged back into the main river. The flow from the weir flows down the channel towards the mid-channel island.

2.2 Single point measurements

The field campaign was split over two days in low flow conditions to cover as much as the localized areas as possible. On 24th August 2015 an extensive spatial survey was conducted over a 6 hour period. Figure 2a) shows the points were near bed flow velocity (5 cm from bed) was measured using a Valeport Electromag netic Flow Meter (EMF), bed elevation was measured using aTrimble RTK GPS, depth was measured using a simple rigid meter rule and phytobenthic biomass was measured using the bbe moldeanke BenthoTorch an in situ fluorometry device. Measurements were recorded in as many points as possible. Deep areas and areas sur rounding concrete structures and large boulders were often difficult to measure meaning that some areas

were not sampled. Following River Habitats Survey

descriptors (Raven et al., 1998) a visual representa

tion of habitats was sketched onto an aerial image of

the site. Figure 3. Interpolated plot of the single point flow velocity measurements below Stockport Hydro (SJ936789441). 2.3 Flow profile measurements On 10th September 2015 velocity profiles where measured at the near and far side river bank (hydro side and non-hydro side) using an Acoustic Doppler Current Profiler (ADCP) anchored in stationary positions for 5 minute time periods. Figure 2b) is a schematic diagram of the points where the ADCP was anchored. An attempt was made to collect an even amount of profiles on the hydro and non-hydro side of the island. Measurement locations were often dictated by depth as the ADCP needs a minimum sampling depth of 20 cm. 2.4 Data analysis Spatial survey data including near bed flow velocities, bed elevation, depth and phytobenthic biomass was interpolated using kriging methods in Surfer Software. Habitat sketches were transferred from sketches to shape files in Surfer Software. Linear regression was used to explore the relationship between phytobenthic biomass and flow velocity. ACDP measurements were averaged over depth and time and interpolated using kriging methods in Surfer Software. Velocity vectors were layered on top of the surface plot to display direction of flow. Surface plots were annotated to define and display distinct hydrological and morphological features. 3 RESULTS 3.1 Single point measurements For the single point measurements the level on the weir remained at 6cm and the abstraction rate was 1.11 m^3/s ±0.02 m^3/s. The distribution of measured flow velocities is shown in Figure 3. Areas of high flow velocity are found between the hydro side deposit

Figure 4. Interpolated plot of the single point depth mea

surements below Stockport Hydro (SJ936789441).

Figure 5. Interpolated plot of the single point bio

film biomass measurements below Stockport Hydro

(SJ936789441).

and the river bank, between the bank deposit and Mid channel Island, and between the non-hydro side deposit

and concrete structure (Figure 3). Lower velocities are

typically recorded on the non-hydro side of the river

(Figure 3) except velocities below the second concrete

structure. Figure 4 shows the measured water depths

clearly indicating zones of erosion and deposition. Figure 5 shows phytobenthic biofilm biomass.

Biomass appears lower on the hydro side of the

river (Figure 5). Linear regression plots show down

ward trends in biofilm biomass as velocity increases Figure 6. Interpolated plot of the single point elevation measurements below Stockport Hydro (SJ936789441). (Figure 9) Low R-squared values do not support linear regression patterns. Figure 6 shows the interpolated Ordinance Survey elevations where dashed lines represent deposits. The elevation plot shows distinct morphological features including erosion and deposition. The island has the highest elevation with measurements up to 54.7 m.The bank deposit, hydro side and non-hydro side deposits have similar elevations. A pool from below the weir extends just above the island (Figure 6).An area of low elevation (high water depth) is evident in the turbine forebay. Figure 7 displays habitat sketches from below Stockport Hydro. Distinct hydrological and morphological features are evident. A weir pool extends towards a glide on the non-hydro side of the river. An area of stagnation is visible at the upstream junction corner of the outlet.An area of separated flow is visible at the downstream junction corner of the outlet. There is an area of separated flow adjacent to the hydro side deposit at the point where the water from the outlet is discharged back into the main channel. A run extends from the outlet of the hydro scheme towards the end of the island. Riffles extend from the bank deposit on the hydro side of the river and from below the mid-channel island. Glides and pools form the majority of the area on the non-hydro side of the river. A run extends from the first concrete structure on the non-hydro side of the river, beyond the second concrete structure towards the island. Riffles are evident between both deposits on the hydro side and non-hydro-side of the river and the mid-channel island. 3.2 Flow profile measurements For the flow profile measurements the level on the weir remained at 6cm and the abstraction rate was

Figure 7. Habitat sketches from below Stockport Hydro

(SJ936789441) based on RHS descriptors.

Figure 8. Annotated interpolated plot of measured flow

velocity below Stockport Hydro (SJ936789441) using the

ADCP.

1.00 m^3/s ±0.03 m^3/s. Interpolated plots reveal areas

of distinct hydrological and morphological features

Figure 8). Separation is evident between the outlets

downstream junction corner and the bank deposit (Fig

ure 8).Another area of separation is evident at adjacent

to the hydro-side deposit extending into the outlet.

Deflection is evident (Figure 8) where the two flows

collide. Deflection is also recorded on the non-hydro side of the river on the opposite side of the midchannel island. A bar was evident below an area of separation at the outlet.Acceleration occurred between the hydro side deposit and the bank and where the channel narrowed between the bank deposit and midchannel island (Figure 8). Acceleration and deflection was evident between the non-hydro-side river bank and deposit. Lower velocities were visible on the nonhydro side of the channel compared to the hydro-side of the channel just as in single point measurements. 4 DISCUSSION The measurements showed a difference between flow velocity and water depth, and to some level biofilm biomass on the hydro and non-hydro side of the river. High flow velocities towards the hydro bank supports the Environment Agencies (Mould et al., 2015) findings at Romney Weir in the River Thames were higher velocities were found on the hydro-side of the channel and lower velocities were recorded on the on the non-hydro side of the channel. There is also clear evidence of distinct hydrological and morphological features typically attributed to the collision of two flows (Figure 3, Figure 7 and Figure 8). Habitat sketches display an area of stagnation at the upstream outlet junction corner (Figure 7). Stagnation is often recorded at the upstream junction corner at confluences (Szuipany et al., 2009) and as such is potentially related to the scheme. Separated flow is evident at the point where the two flows collide. There is clear evidence of a deposit below this area of separation (Figure 3, Figure 7 and Figure 8). It is possible that this deposit is a result of sediment passing through the turbine channel from upstream. This would match the draw down theory presented by Anderson et al., (2014). As the flow separates and the velocity reduces in the separation zone the material is potentially deposited. A second area of separated flow is evident at the downstream junction corner of the outlet (Figure 3, Figure 7 and Figure 8). This is another feature which is typically found were one channel meets another (Szuipany et al., 2009) and as such is potentially related to the flow discharging from the outlet. Below this area of separated flow is a bank deposit. Bank deposits are typically associated with the area of separation at confluence channels and as such could be related to the outlet of the hydro scheme. Figure 8 shows deflection of flow in the main river channel at the point where the outlet discharges water. This deflection is likely to have been caused by the hydro scheme.This is a feature which is typically associated with the interface of two flows (Szuipany et al., 2009). It must also been noted that the highest velocities are found at the points were the channel narrows (Figure 3 and Figure 8). There are areas of high velocity on both sides of the channel although the hydro side of the channel typically has the highest overall velocity.

Figure 9. Correlation between single point flow velocity

and biofilm biomass below Stockport Hydro (SJ936789441).

High velocities are evident between the hydro deposit

and the river bank, the bank bar and the island and the

non-hydro side deposit and concrete structure (Fig

ure 3 and Figure 8). This is not surprising considering

that acceleration is typically associated with channel

narrowing.

The main morphological features, the scour pool

and the mid channel island, are features typically asso ciated with the aquatic environment below weirs. The water which cascades over the weir causes scouring of the river bed often creating a large scour pool (Mould et al., 2015). Were the energy in the flow is reduced the scoured material is often deposited forming an island or bar (Mould et al., 2015).This suggests that the scour pool and mid-channel island found below Stockport Hydro was a feature in the channel before the scheme was installed. Figure 5 shows biofilm biomass below Stockport hydro. While there appears to be lower biomass on the hydro-side of the river, which in theory could be related to the scouring effect of the high flows from outlet (Law, 2011), there is no relationship between biofilm biomass and flow velocity (Figure 9).The phy tobenthic biofilm can change according to a number of variables including depth, flow velocity, predation, light penetration, temperature and pH (Law, 2011). An investigation into the combined effects of a num ber of variables might yield better results and will be considered in future analysis. Equally analysis of communities' on the hydro and non-hydro side of the river will be carried out to determine if changes in flow distribution cause ecologically significant impacts.

5 CONCLUSIONS

Spatial surveys have been used to identify hydrolog

Paish, O. 2002. Micro-Hydropower: Status and Prospects. Proc. Inst. of Mech. Eng. A J. Power Energy 216 (1): 31-40.

Raven, P.J., Holmes, N.T.H., Dawson, F.H., Fox, P.J.A., Everard, M., Fozzard, I.R. & Rouen, K.J. 1998. River Habitat Quality the physical character of rivers and streams in the UK and Isle of Man. River Habitat Survey Report NO. 2.Available at

Robson, A., Cowx, I. and Harvey, J. 2011. Impact of run of the river hydro schemes on fish populations. Scotland and Northern Ireland Forum for Environmental Research (SNIFFER). Available at http://www.sniffer.org.uk/

New concepts in small hydropower plants schemes in Romania

F. Popa

ISPH Project Development, Bucharest, Romania

D. Florescu

Individual Hydropower Consultant, Ploiesti, Romania

B. Popa

University Politehnica of Bucharest, Bucharest, Romania

ABSTRACT: In Romania, at the end of the communist era, the hydropower schemes were over dimensioned

in terms of installed capacity, hence in hydraulic structures and equipment as well. Today, more than 100 small

hydropower plants (hydropower plants with installed capacity less than or equal to 10 MW) must be refurbished

until the end of 2016 in order to benefit from the E-RES support system. According to the existing regulations,

the refurbished SHPPs receive 2 green certificates per MWh produced, which can be sold for more than 27 Euros

each. This is the only chance for these plants to become profitable. The existing hydropower schemes must be

analysed so as to determine whether they were adequately dimensioned and whether the river sector was fully

developed. In order to use the existing assets as much as possible, to fully develop the partially developed river

sectors and to correct the deliberate designing errors, new concepts must be applied for the refurbishment stage.

The paper presents some of these concepts and case studies for illustration.

1 INTRODUCTION

From historical perspective, Romanian hydropower

begins with small plants (SHPPs). The first SHPP,

built in the last part of the nineteenth century, had an

installed capacity below 1 MW; it was used as local

power supply, being equipped with Automatic Speed

Regulators, even for capacities below 100 kW, and,

frequently, with Automatic Voltage Regulators, in the

absence of a National Power Grid. The adopted devel

opment schemes solutions were mainly run of river and

they functioned on the flowing river as there were few

ponds. At the end of year 1900 there were 19 SHPPs

having a total installed capacity of 4.115 MW and an

average energy output of about 16.26 GWh/year. In 1950, 115 SHPPs are registered having a total

installed capacity of about 22 MW and an average out

put of about 90 GWh/year. They were economic units

of small dimensions.

The first electrification plan of the country recorded

the realization of a powerful network of large hydropower plants (HPPs) as well as a large number of SHPPs connected to the national power grid through local networks. Today, the majority of these SHPPs are decommissioned because the development of the national power grid has led to the giving up of their modernization and overhaul.

A very important period, called the national capi talization program for the small hydropower potential in Romania, will be described in a later paragraph, includes the ninth decade of the past century when a vast and coherent program was applied to this sector. The events at the end of 1989, the changing of the communist regime with a capitalist one, practically marked the cessation of this program. Only sporadic commissioning of such type of power objectives (begun before 1990) was seen until 2006. The last important period started in the year 2006 and represents a significant qualitative improvement in the conceptual approach to SHPPs complex issues in the new context of the market economy and in the technical solutions adopted for construction and electromechanical equipments. At the end of the communist era, Romania had overdimensioned hydropower schemes (installed capacity, hydraulic structures and equipment). An energy sector overview in Romania, with the support scheme for RES (RES-E) produced electricity, as well as the situation of RES energy potential, 2011 RES-E production, share of RES-based plants in the total energy mix in 2010, the RES-E produced by hydropower plants in 2010, they all are presented in Colesca & Ciocoiu (2013). Mandatory national targets for the energy share from renewable sources in the final energy consumption for year 2020 in Romania is 24%. According to Romanian legislation, a SHPP is considered a hydropower plant if it has the installed capacity of maximum 10 MW; refurbished SHPPs receive 2 green certificates per MWh produced, which can be sold for more than 27 Euros each. At the end of 2015 (Popa, in prep., Transelectrica, 2016) there were 386 SHPPs, with installed capacity of 585 MW and mean annual electricity production of about 2TWh. Today, more than 100 SHPPs must be refurbished and rehabilitated until the end of 2016, in order to benefit from the E-RES support system. It is the only chance for these plants to become profitable. The existing hydropower schemes must be anal ysed so as to determine whether they were adequately dimensioned and whether the river sector was fully developed. In order to use the existing assets as much as possible, to fully develop the partially developed river sectors and to correct the deliberate design ing errors, new concepts must be applied for the refurbishment stage. The main subject of this paper is to present current concepts and technical solutions to be applied to the existing SHPP schemes and study cases.

2 BIBLIOGRAPHICAL SURVEY

Many scientific papers have been written on the sub ject of new concepts for small hydropower (SHP) schemes in the world, from the perspectives of the design, site, mechanical or electrical equipment, and even operation. Such papers address either the newly developed SHPPs or the ones that are to be refurbished or rehabilitated. Under all circumstances, those new concepts must take into account the latest research and development (R & D) advancements in the related domain and also the policy on renewable energy sources (RES) in the country where the development is performed. Regarding SHP potential and the possible further development of SHPPs in different countries, there are papers that show the distribution of the SHP resource in the world (Ellabban et al. 2014). These papers also address aspects as power electronics and facilities of smart grids for connection to new RES based plants. Sternberg (2010) shows the importance of hydropower all over the world in terms of resource, instrument of change, environment, geopolitics and future development. To have an idea on how countries with huge hydropower potential and development regard the present and the sustainable future development in a world looking for environment care and protection, papers as Kong et al. (2015) for China, Westin et al. (2014) for Brazil can be consulted. Other papers, as Lehner et al. (2013), are dedicated to the state of the art of hydropower in Europe, with emphasis on aspects such as: already developed hydropower potential, evaluation of gross hydropower potential, without taking into account the climate change, and using the WaterGAP model, stating that the evaluation of other types of potentials requires additional data and information. With regard to a specific European country profile on SHPPs, many papers are dedicated to presentations

of the potential, policy, and development, such as: Kaldellis (2006) for Greece, Montes et al. (2005) for Spain, Ostojic et al. (2013) and Panić et al. (2013) for Serbia, Stritih et al. (2006) for Slovenia, Zimny et al. (2013) for Poland, in a more complex approach, i.e. in the framework of showing directions in hydropower development worldwide and in Europe for the period 1995–2011. Papers as Paish (2002) and Okot (2013) on technologies should be cited, as they presented a review and the current status of small hydropower development schemes and technologies. Kumar & Singal (2015a) analyse problems related to SHPPs operation and maintenance from turbines perspective. After presenting different types of turbines, they introduced some operational problems as: cavitation, erosion, fatigue, material defects.They also presented suggestions for remedial measures. A comprehensive literature review on refurbishment and rehabilitation of hydropower plants was performed by Rahi & Chandel (2015). Loots et al. (2015) reviewed the technologies that exist in South Africa for low head SHPPs. They also addressed possible applications for existing dams, rivers and irrigation systems, industrial and urban discharge, storm water systems and water distribution networks. Kusakana (2014) presented other innovative technologies which could be applied in South Africa to enable SHPPs development in terms of costs and reliability, as a viable option for rural electrification.The survey of technologies regarded: penstocks, turbines, kinetic devices, generators and controllers. In conclusion, certain guidelines were elaborated for selecting adequate design and equipment. Yu & Xu (2016) studied a problem that relates to both existing and prospective hydropower plants, the problem of the so-called environmental or ecological flow. Issues addressed in this paper include the compensation flow, the achievement modes, the existing mechanisms and the future trends. Other topics are related to SHPPs integration into existing hydro-technical systems: Almeida et al. (2011) described a project for the integration of a SHPP into a multi-purpose dam bridge. Manders et al. (2016) showed SHPP builders'issues and strategies for developments in the Netherlands. They presented case studies which addressed subjects like location, fish passes, stakeholders' interest, political support and public perceptions. Many papers analyse and evaluate the design of a new SHPP or on the verification of the appropriate dimensioning for existing SHPP so as to determine refurbishment and/or rehabilitation necessities. Many recent articles deal with

SHPPs issues from technical and economic angles: optimization for location selection, development scheme design, electromechanical equipment options, optimization of operation and grid integration. Mishra et al. (2011) made a research review on aspects of optimal SHPPs installation: technology, simulation models, economic analysis and costs. Regarding site selection, Rojanamon et al. (2009)

applied the geographical information system (GIS)

taking into account the technical, economic, environ

mental and social criteria. Kumar & Singal (2015b)

applied the Multiple Attribute Decision Making

(MADM) method for assessing some existing SHPPs

in order to select the best operating one. Sachdev et al. (2015) made a bibliographical sur

vey on SHPPs analysis and evaluation, touching

aspects like: development, design, distributed gener

ation, mathematical modelling of SHPPs, technical

analysis and economics, as well as control.

Ardizzon et al. (2014) presented a new generation

of SHPPs together with pumped storage plants (PSPs),

addressing issues like: optimal sizing, variable-speed

technology, optimal operating strategies, computa

tional fluid dynamics (CFD) as tool for improving

mechanical equipment performances and for design

ing new runners for rehabilitating old plants. Ogayar & Vidal (2009) determined costs of SHPP

turbines and generators in different alternatives. In a more comprehensive manner, Ogayar et al.

(2009) performed an analysis of the cost required by

SHPPs refurbishment. The addressed issues related to hydro-technical works: dam, intake, pipeline (chan nel, tunnel, forced), forebay, penstock (steel or glass reinforced pipe – GRP), powerhouse and the electro mechanical equipment, protection, regulation and con trol as well as network connection and the transmis sion line.

3 FAULTS IN SHPP DEVELOPMENT

3.1 The national capitalization program for the small hydropower potential in Romania

SHP potential in Romania is significant. In part, it was already used, but incorrectly or insufficiently ex ploited because of the technical, economic and politi cal constraints of the ’80s. After changing the political regime, this situation raised problems in defining and updating concepts and technical solutions intended to improve the capitalization of the useful potential of those rivers, already developed for hydropower, and to streamline the existing SHPPs.

A vast capitalization program for the small hydro power potential in Romania was initiated at the begin ning of the ’80s by constructing SHPPs connected to the existing 0.4 kV and 20 kV networks. Unfortunately, this program had the same centralized character as the political system of the country at the time.

As a result, the establishment and the parameter ization of future SHPPs locations, defining arrangement solutions and designing constructions as well as electromechanical equipment, suffered from those constraints imposed by the mandatory standardization in the most limited number of options.

To address this issue, the Institute for Hydropower Studies and Designs (IHSD, today ISPH Project Development) was designated to coordinate the identification of the developable hydropower potential and of the locations fitting the pre-established criteria; it elaborated a 'Catalogue of Projects and Constructive Type Solutions'. Similarly, coordinated by Resita Machines Construction Plant which manufactured power machines and equipment, a limited number of standardized hydropower units (HPUs), turbines and generators were designed and produced. A function of location parameters, equipping these SHPPs was established in 'The Selection Diagram for Microturbines', jointly elaborated by the two coordinating entities. In practice, this faulty conceptual approach led to identification and incomplete capitalization of SHP potential both quantitatively (reduced sectors of water flows) and qualitatively (diminished power productions especially due to reduced efficiency and reduced number of operation hours in the hydropower units). Besides the technical disadvantage stemming from standardization constraints, we also have the imperative of meeting certain imposed technical and economic indicators, formally achieved using some overequipping coefficients justified by the aspect of the flow duration curves (FDC). In this way, the difference between the calculated values and the real data subsequently obtained by the actual hydropower objectives increased even more. Without going into statistical details, we need to point to the fact that the electrical power production obtained in the almost 200 SHPP built in this program, achieved, on average, less than 70% of its project value during the first years of operation and followed a constantly descending trend afterwards. The analysis of each SHPP shows significant differences, from 100% achievement of project electricity production to 20%, which results in the decommissioning of the plant in a very short time. 3.2 Rules for standardization Selection of SHPPs arrangement pattern and building solutions was accompanied by recommendations on project frameworks adaptation (via technical execution documentation)to the actual conditions on the sites approved by geologists for the general stability of the location in point of foundation conditions. Straight areas of river bed, as narrow as possible, were preferred for the intake of the hydropower development (HPD) with tall shores. The mandatory location for the intake was on the convex shore of the river bed. Rocky foundations were preferred and damming works on the shores were executed when the elevation of the shores was less than +4 m. The chosen location had to allow the application of one of the four solution frameworks presented in the catalogue: two rigid solutions for rocky terrains and two elastic solutions for non rocky terrains. The HPD solution included a mandatory head pond whose purpose was to ensure turbines operation at the installed flow level therefore, at the optimum efficiency level. The following rules were imposed for ponds: The active capacity allows the operation of a unit for at least half an hour at the installed flow. The chosen location is situated outside the river bed,

in an area not affected by floods, as close to the intake as possible. The chosen location has a favourable con figuration of the terrain (stretching) that allows the achievement of the necessary water volume. The damming solution is imposed in alluvial ter rains with slopes less than 20%. The stone masonry solution is imposed for rocky terrains with slopes less than 30%. According to local conditions, to evacuate the deposits from the pond, washing and evacuation equip ment is provided. As an alternative, the dry evacuation method (digging and transport) can be used to evacuate the deposits. Three constructive solutions were imposed for diversions: free flow channel, penstocks (pressurized pipes), and the mixed solution. The free flow channel was considered the basic solu

tion. Penstock adaptation needed justifications and

economic efficiency calculations fitting the situation

into one of the 5 cases presented in detail. The construction solution for projects type with

shore canals was clay or stone. The building solution

for pressurized pipes was the concrete pipe. As far as the route is concerned, the location was

chosen to be along the existing valley road, away from

the action of floods or landslides, without affecting

agricultural and forest lands. The solutions chosen for the powerhouse had to be

consistent with the approved projects. As far as equip

ping is concerned, as a general rule, the installation of

a single hydropower unit (HPU) was imposed for the

corresponding projects types presented. The equipping with multiple HPUs was admitted in

well-defined cases, as follows:

– the installed power (capacity) exceeded the capacity of the largest HPU indicated in a certain “Selection Diagram for Micro-turbines”;

– the head pond could not be done and, based on power and economic calculations, two or more units of unequal power (1/3 and 2/3 from the total installed capacity of the HPP) were imposed. The verification of the head splitting option was

imposed for either exception so that more power plants

equipped with a single HPU were obtained. Thus, all the SHPPs built in the 80s were affected

by dimensioning and design faults. In 2002, enforcing GD 554 (2002), 279 HPPs total ising an installed capacity (Pi) of 451 MW (Table 1)

were included in the patrimony of Hidroelectrica,

which was put in charge of preparing the task book

and selling them by public auction. The designed energy output for those HPPs is

1328 GWh/year, with a mean of 2945 h/year duration

of installed capacity use, which places them among

peaking power plants. Borbely (2016) presents the phases in which a part

of the SHPPs were sold. Thus, 87 of them were sold

between 2004 and 2008 and 36 were sold between

2013 and 2015. In the forthcoming period, 20 SHPPs Table 1. HPPs included in the patrimony of Hidroelectrica in order to be sold. Number Pi Pi per SHPP (MW) – [MW] Pi below 1 MW 187 82 Pi between 1 and 10 MW 83 249 Pi above 10 MW 9 120 Total 279 451 grouped in 14 packages of assets are expected to be available for sale, totalling 16.5 MW installed capacity and 49.2 GWh design energy output. This will be followed by other assets estimated to be additional 30...40 SHPPs totalling approximately 30 MW installed capacity and 100 MWh designed energy output. Brodina HPD was acquired from Hidroelectrica via public auction in 2006 by a private investor that launched a vast program for the refurbishment of existing plants and redesigning of the non-developed sectors. Having this SHPP as an example, we will present new concepts applied for the refurbishment and redesign of the development scheme as a whole. 4 APPLIED CONCEPTS AND SOLUTIONS 4.1 Principles The general principle is the determination of the development solution for integrating the existing arrangement, able to ensure the integrated capitalization of the hydropower potential for each catchment. A system of hydropower plants in cascade arrangement is functionally linked by the main hydraulic wire, having only one main intake and several secondary intakes intended to optimize the utilization of the potential of the tributaries located in the catchment of the developed river and to supplement the flows produced by the catchment difference. The solution for evacuating the produced electrical power implies individual physical connections for each production unit. However, delimitations of administration with the distribution network administrator as well as implications of the network injections are analyzed unitarily. Distance management is accomplished by a single set of automated cooperating applications which acquire, process, circulate, present and record the data characterizing the process of generation and commercial capitalization of the produced electricity. In the following paragraphs we will discuss the (a) to (h) solutions that were proposed for application on the existing schemes. Their application is exemplified on Brodina HPD and on four other hydropower developments in section 4.2. a.The optimum determination of the developed sector upstream through the correct positioning of the

main intake, so that alluvial contribution is diminished

and the operating regimes (pre-established according

to the achievement of the technical and economic

efficiency indicators) are respected. b. The rehabilitation and refurbishment of head

works downstream the main new intake by: partially

maintaining their functional role but sizing them in

such a way that it processes the flows resulting from

the catchment difference and the supplementation of

the role of the de-silting tank/head pond in the stilling

basin through partial incorporation into the infras

tructure of a new hydropower plant which directly

capitalizes the main intake potential. c.On the other hand, we specify that one of the solu

tions imposed in the previous step was the Tyrolean

intake and the pond, both components being linked

together by a pipe. In time, this solution determined

the severe reduction of the water processing capacity

because of the sand and gravel warping, including the

total unavailability during floods.

The newly proposed solution is building a lateral intake to integrate the existing intake front and ensure direct processing of collected flows in the pond. d. Re-equipping the existing hydropower plants with modern HPUs, capable of continuous processing at quasi-constant efficiency of tributary flows situated between 30%–100% of the installed flow. Fitting new equipment meant substantial infrastruc ture alteration and, at times, impossibility of perform ing necessary modifications due to dramatic changes in the static and dynamic resistance of the construction. Hence a simple solution was proposed, which meant the construction of a hydropower plant to integrate the existing stilling basin by correlating the horizontal and the vertical dimensions with the recommenda tions of the equipment supplier. The advantages of the proposed solution are significant:

- it provides, at the least, maintaining of initial HPUs parameters,
- it gives the construction advantage of a hydropower plant building integrally adapted to the recommen dations of the equipment supplier,
- it reduces construction costs to less than 50% of what was required by a new hydropower plant with its whole infrastructure,
- it allows the choice of an architectural solution spe

cific to the area where the HPD is located, enabling

natural integration into the local landscape. e. The by-pass pipes and valves solution: choos

ing the by-pass solution sized at one third (1/3) of the

installed flow grants the continuous flow of water on

the main hydraulic wire (protection against freezing)

during accidental HPU shutdown as well as in inter

vals of low flows, under the turbine operational limit.

Situations like these are frequently encountered during

the cold periods of the year. On the other hand, during an accidental shutdown

of a hydropower plant, the by-pass insures the partial

operation of the downstream HPUs, hence reducing

losses for the HPD as a result of its unavailability. f. The integral capitalization of the hydropower potential, within the upstream and the downstream limits of the existing development scheme. This idea means building some SHPPs on the river sectors that are not yet developed with the following structure: loading chamber at the outlet of the upstream hydropower plant, headrace, the powerhouse evacuating in the stilling basin, which is the loading chamber of the downstream SHPP. g. The optimum determination of the downstream limit of the development solution by building new power units, justified both technically and economically, through the transformation of the last existing hydropower plant into the loading chamber of the new power units built downstream. h. The supplementation of the processed inflows by the contribution of some secondary intakes correspondingly positioned on tributary rivers situated in the same catchment. In this situation, the hydraulic calculations performed for every operating regime on the main hydraulic wire are very important in order to determine the elevation of these intakes. 4.2 Case studies According to the initial solution, the hydropower development on river Brodina, (consisting in Ehreste, Brodina 1 and Brodina 2 SHPPs), was accomplished during the ’80s. These three power objectives were commissioned in 1985, 1984 and 1983, respectively. In Romanian legislation (ANRE Order 48, 2014) the meaning of refurbishment for hydropower plants includes: a) operations of drive replacement for one or more power units of the turbine rotor and/or nozzles intake or unit director and/or valves intake with components that have not been used to produce electricity; and/or b) execution of hydraulic works in the whole hydropower development, hydro modernization, investment in hydro equipment, electrical equipment, automation, communication and security. In order to be considered refurbishment, the proof of completion of refurbishment operations provided above is based on: reception minutes attesting the completion of the works set out in the feasibility study or in other studies that triggered refurbishment operations; reception minutes for commissioning; the report by an independent auditor confirming that the accountable value of assets related to SHPP after refurbishment is above 1,500 Euros/kW installed. The original situation and the current one for HPD Brodina are shown in Figure 1. For building the existing scheme, the principles presented in section 4.1, (a) to (h), were applied in order to correct the concept and design faults presented in section 3. The same principles were applied for other existing hydropower developments. Figures 2–5 show the present situation, after refurbishment and further development, for HPD Boia, HPD Sebes, HPD BarsaGhimbasel-Bran and HPD Tarlung, respectively.

Figure 1. Brodina hydropower development. Left side: orig

inal situation, right side: present situation; (a)–(h) applied

concepts.

5 CONCLUSIONS

SHP potential in Romania is significant but only about

half was already used; it was exploited incorrectly or

insufficiently because of the technical, economical and

political constraints of the ’80s. The paper presents the constraints and highlights

new concepts and technical solutions intended to

improve the capitalization of the useful potential.

Some successful case studies were presented demon strating that it is possible to accomplish a fully

hydropower developed river by correcting all the faults Figure 2. Boia hydropower development. Figure 3. Sebes hydropower development. Figure 4. Barsa-Ghimbasel-Bran hydropower development. Figure 5. Tarlung hydropower development.

in conception, design and constructions by applying

new and appropriate concepts.

Almeida, A.T., Carlos Invernoa, C., de Almeida, J.L., Sâ Marquesb, J.A., Santosa, B. 2011. Small-hydropower integration in a multi-purpose dam-bridge for sustainable urban mobility. Renewable and Sustainable Energy Reviews, 15: 5092–5103.

ANRE Order 48/2014. Regulation regarding the accreditation of producers of electricity from renewable energy sources for applying the system of green certificates promotion.

Ardizzon, G., Cavazzini, G., Pavesi, G. 2014. A new generation of small hydro and pumped-hydro power plants: Advances and future challenges. Renewable and Sustainable Energy Reviews, 31: 746–761.

Borbely, K. Hidroelectrica for a new beginning. Romanian HydroPower Energy Summit, Bucharest, February 25, 2016.

Colesca, S.E. & Ciocoiu, C.N. 2013. An overview of the Romanian renewable energy sector. Renewable and Sustainable Energy Reviews, 24: 149–158.

Ellabban, O., Abu-Rub, H., Blaabjerg, F. 2014. Renewable energy resources: Current status, future prospects and their enabling technology. Renewable and Sustainable Energy Reviews, 39: 748–764.

GD 554/2002. Transition of hydropower plants from the commercial company of electricity and heat production “Termoelectrica”, from the company for electricity distribution and supply “Electrica” and the national society “Nuclearelectrica” to the patrimony of commercial company of electricity generation “Hidroelectrica”.

Kaldellis, J.K. 2006. Critical evaluation of the hydropower applications in Greece. Renewable and Sustainable Energy Reviews, 12: 218–234.

Kong, Y., Wang, J., Kong, Z., Song, F., Liu, Z., Wei, C. 2015. Small hydropower in China: The survey and sustainable future. Renewable and Sustainable Energy Reviews, 48: 425-433.

Kumar, R., Singal, S.K. 2015a. Selection of best Operating Site of SHP plant based on performance. Procedia Social and Behavioral Sciences, 189: 110-116.

Kumar, R., Singal, S.K. 2015b. Operation and Maintenance problems in hydro turbine material in small hydro power plant. 4th International Conference on Materials Processing and Characterization. Materials Today: Proceedings 2: 2323 - 2331.

Kusakana, K. 2014. A survey of innovative technologies increasing the viability of micro-hydropower as a cost effective rural electrification option in South Africa. Renewable and Sustainable Energy Reviews, 37: 370-379.

Lehner, B., Czisch, G. Vassolo, S. 2013. EuroWasser: Europe's hydropower potential today and in the future.

Loots, I., van Dijk, M., Barta, B., van Vuuren, S.J., Bhagwan, J.N. 2015. A review of low head hydropower technologies and applications in a South African context. Renewable and Sustainable Energy Reviews, 50: 1254-1268.

Manders, T.N., Hoffken, J.I., van der Vleuten, E.B.A. 2016. Small-scale hydropower in the Netherlands: Problems and strategies of system builders. Renewable and Sustainable Energy Reviews, 59: 1493-1503. Mishra, S., Singal, S.K. & Khatod, D.K. 2011. Optimal installation of small hydropower plant - A review. Renewable and Sustainable Energy Reviews, 15: 3862-3869. Montes, G.M., Lopez, M.M.S., Gamez, M.C.R., Ondina, A.M. 2005. An overview of renewable energy in Spain. The small hydro-power case. Renewable and Sustainable Energy Reviews, 9: 521-534. Ogayar, B. & Vidal, P.G. 2009. Cost determination of the electro-mechanical equipment of a small hydro-power plant. Renewable Energy, 34: 6-13. Ogayar, B., Vidal, P.G., Hernandez, J.C. 2009. Analysis of the cost for the refurbishment of small hydropower plants. Renewable Energy, 34: 2501-2509. Okot, D.K. 2013. Review of small hydropower technology. Renewable and Sustainable Energy Reviews, 26: 515-520. Ostojic, G., Stankovski, S., Ratkovic, Z., Miladinovic, L. 2013. Development of hydro potential in Republic Srpska. Renewable and Sustainable Energy Reviews, 28: 196-203. Paish, O. 2002. Small hydro power: technology and current status. Renewable and Sustainable Energy Reviews, 6:

537–556. Panic´, M., Urošev, M., Pešic´, A.M., Brankov, J., Bjeljac, Ž. 2013. Small hydropower plants in Serbia: Hydropower potential, current state and perspectives. Renewable and Sustainable Energy Reviews, 23: 341–349. Popa, B., Popa, F. & Tica, E.I. Small hydropower in Romania. In preparation. Rahi, O.P. & Chandel, A.K. 2015. Refurbishment and uprating of hydropower plants—A literature review. Renewable and Sustainable Energy Reviews, 48: 726–737. Rojanamon, P., Chaisomphob,T., Bureekul,T. 2009.Application of geographical information system to site selection of small run-of-river hydropower project by considering engineering/economic/environmental criteria and social impact. Renewable and Sustainable Energy Reviews, 13: 2336–2348. Sachdev, H.S., Akella, A.K., Kumar, N. 2015. Analysis and evaluation of small hydropower plants: A bibliographical survey. Renewable and Sustainable Energy Reviews, 51: 1013–1022. Sternberg, R. 2010. Hydropower's future, the environment, and global electricity systems. Renewable and Sustainable Energy Reviews, 14: 713–723. Sternberg, R. 2010. Hydropower's future, the environment, and global electricity systems. Renewable and Sustainable Energy Reviews, 14: 713–723. Stritih, U., Zupan, G., Butala, V. 2006 (in press). Review of green electricity production in Slovenia. Renewable and Sustainable Energy Reviews. Transelectrica, www.transelectrica.ro, accessed March 2016. Westin. F.F., dos Santos, M.A., Martins, I.D. 2014. Hydropower expansion and analysis of the use of strategic and integrated environmental assessment tools in Brazil. Renewable and Sustainable Energy Reviews, 37: 750–761. Yu, B., Xu, L. 2016. Review of ecological compensation in hydropower development. Renewable and Sustainable Energy Reviews, 55: 729–738. Zimny, J., Michalak, P., Bielik, S., Szczotka, K. 2013. Directions in development of hydropower in the world, in Europe and Poland in the period 1995–2011. Renewable and Sustainable Energy Reviews, 21: 117–130. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Fish behavioral and mortality study at intake and turbine

P. Rutschmann & S. Schäfer

Hydraulic and Water Research Engineering, Technische Universität München, Germany

B. Rutschmann

HOBOS Team, University of Würzburg, Germany

F. Geiger

Hydraulic and Water Research Engineering, Technische Universität München, Germany

ABSTRACT: Over the last three years, tests with almost 2000 fishes have been conducted at the patented

TUM Hydro Shaft Powerplant involving five different species, namely trouts, graylings, barbels, bullheads and

minnows with fish sizes between 50 and 200 mm. The turbine mortality very much differed among the tested fish

species, and a large variation is detected over the various test runs. Amazingly the worst swimmer, the bullhead,

showed the lowest probability of damage or injury in the turbine. However, this contribution analyses the results

species independently and shows that probabilities of mortality or injury for given bar clearance depend on the

intake flow velocity, the operation of the turbine and the fish length. The present paper discusses the various

influences of the hydraulic conditions at the trash rack and the operational setup of the turbine on mortalities

and tries to quantify these effects. Also the paper separates the size dependent probability of trash rack passage

from that of pure turbine damage and thus allows to derive total downstream passage survival rates for the Shaft

Powerplant.

1 INTRODUCTION

In 2009 the so called TUM hydro shaft concept

has been invented at the Chair of Hydraulic and

Water Resources Engineering of Technische Univer

sität München (TUM), Germany.The patented concept

is well suited to use the energy of natural and artificial steps in rivers and consists of a shaft which sits in the river bed, a horizontal trash rack with cleaner parallel to the river bed, a sub-surface turbine with perma nent magnetic generator followed by a suction pipe to release the water into the downstream. Later, besides a few accompanying patents, a second main patent fam ily has been applied for, the so called TUM multi-shaft concept, which uses several shafts and combines them with integrated eco-migration corridors in the axis and on both sides of a river. Behind the TUM hydro shaft power plant (TUM HSPP) there are a few ideas related to economy and ecology which are explained in the following (see also Figure 1):

1. The plant has a very simple geometry and thus, is rapidly planned and built with minimum costs.

2. It is relatively cheap and easy to extend the crosssectional area of the intake, thus reducing the intake velocities and enabling fish to freely swim over the rack. 3. At the downstream end of the intake section there is a gate in which bypass openings can be provided, enabling fish an efficient downstream migration path. 4. Sedimentation of the river bed occurs only up to the intake of the plant and additional sediment is either flushed through the turbine or diverted over the trash rack into the downstream. No reservoir sedimentation occurs. 5. The plant is not visible and does not produce noise. 2 TESTS 2.1 The pilot facility In order to investigate the hydraulics and the sediment issues as well as the behavior of fish, and in order to technically develop the concept, a 35 kW pilot plant was built at the Oskar von Miller Institute, the Hydraulic Laboratory of TUM, in Obernach, Bavaria. The TUM-HSPP pilot concept was equipped with a conventional Kaplan turbine of 750 mm diameter and spinning with 333 rpm. As a peculiarity it only has a permanent

magnetic generator. Besides, the plant was equipped with a specifically developed trash rack bar profile and a newly developed underwater trash rack cleaner of Muhr company, Brannenburg, Germany.

Figure 1. Pilot plant and test site of the TUM-HSPP. The outflow at the gate serves as downstream migration bypass. For better understanding see also Figure 2.

Figure 2. Pilot test installation with upstream and down stream basins to release and collect fish. For better under standing see also Figure 1.

The pilot plant used water from the Isar River diverted over a weir into the Lab and measured in a mea suring flume with a calibrated Thomson wear (see Figure 1). In general, very smooth flow conditions could be observed at the intake and visual and acoustic observations can hardly detect operation of the plant. Very precise discharge measurements with the Thomson-weir allowed to determine the maximum efficiency of the plant. The efficiency, defined as the ratio of measured output at the clamps to theoreti cal hydraulic potential, is 87% which is an excellent and very convincing value! Flushing tests with sed iments showed that sediment could be flushed over the trash rack or through the turbine and that no sed iment deposition occurred nor in the upstream nor in the downstream of the suction pipe.

2.2 Hydraulic tests

The behavior of fish for three differing flow condi

tions at the trash rack was investigated. Namely, the

maximum flow velocity at the rack was set to 0.3 m/s,

0.4 m/s and 0.5 m/s in average. This has been achieved Table 1. The investigated 3 differing hydraulic scenarios. Bypass Position Hydraulic Parameters top bottom top bottom top bottom v max-screen [m/s] 0,300 0,400 0,500 h overflow [m] 0,054 0,066 0,073 Q channel [m 3 /s] 1,080 1,410 1,410 1,500 1,640 1,730 Q turbine [m 3 /s] 0,960 0,960 1,280 1,280 1,500 1,500 Q bypass [m 3 /s] 0,080 0,150 0,080 0,150 0,080 0,150 Q gate [m 3 /s] 0,120 0,200 0,130 0,220 0,140 0,230 Figure 3. The two investigated bypass configurations, a surface near (top) and a bottom near (bottom) opening in the gate. by reducing the turbine discharge accordingly. Besides the actual turbine discharge also the flow over the gate and the bypass flow are shown in Table 1. The investigations with fish were run under constant turbine operation conditions. The maximum velocity considers a maximum velocity at the trash rack of 0.5 m/s according to DWA (2014). The minimum velocity is deduced from Ebel (2013). In order to perform ethohydraulic tests of the behavior and damage of fish at the hydropower intake and during downstream migration the test site was equipped with an upstream, and two downstream basins separated by perforated steel sheets in order to avoid fish to leave the test site. The setup is illustrated in Figure 2. Additionally, four underwater cameras were used to observe behavior of fish at the rack. Two differing configurations for downstream migration of fish were investigated, namely a bottom near and a surface near configuration (Figure 3).

Table 2. The investigated 3 differing hydraulic scenarios.

Fish species	Number of fish [-]	Averaged body length [mm]	Standard deviation [mm]
Brown trout	787	144	45
Grayling	733	143	44
Barbel	63	99	51
Minnow	44	81	8
Bullhead	252	81	14

Tests with fish followed the following pattern: The Fish, usually captured in wild rivers, were brought to the Lab at least 48 h in advance in order to adapt them to the local water temperature. For the tests they then were released into the upper basin, and the test run then over a period of 24 h. Fish intending to migrate to the downstream had two options: one over the pro vided bypass, the other through the turbine. Fish that migrated over the bypass ended up in the middle of the three basins (colored in green in Figure 2), and fish that migrated through the turbine ended up in the basin downstream of the suction pipe basins (colored in red in Figure 2). Hourly, during the tests and at the end, all fish were taken off the water with a catcher from all basins and then they were counted. Fish that migrated through the turbine and survived were under observa tion for another 96 h in order to consider secondary or inner damage. Ethohydraulic tests were conducted with five dif ferent species of fish of different sizes. The following fish species and fish numbers were inserted into the upstream basin:

3 RESULTS

3.1 Overview

Tests have been conducted with exactly 1879 fish, of which 38 fish were killed at the turbine and another 10 were more or less seriously hurt. They will subse quently also be counted as “killed”. 670 fish migrated from the upstream into the downstream without being in any way forced to do so, and from these 393 (59%) used the provided bypass through the downstream gate of the HSPP and the other 277 (41%) traversed the turbine. When considering only the bottom near ver sion of the bypass configuration more than 2/3 of fish used this bypass configuration. In the following a more detailed evaluation of the measured results is performed.

3.2 Detailed analysis

About 2/3 of the fish migrated from the upstream into the downstream over the provided bypass with the bot tom near bypass configuration having a higher degree

of acceptance. The probability of taking the passage Figure 4. Probability of bypass passage dependent on fish length and velocity in the intake section. through the turbine was higher for small fish, contrarily the probability of damage during turbine passage was higher for larger fish. As a summary in the very small turbine (750 mm diameter, 333 rpm) in average about 7% of the fish were killed. For prototype plants with larger turbines and lower rotational speeds mortalities below 2% can be expected. The tests showed that the fish in general tried to avoid entrance through the trash rack into the turbine due to the unfamiliar flow situation. Therefore, the horizontal trash rack very much acted as a behavioral barrier. Underwater cameras allowed to observe that fish had no difficulty to swim over the intake at the trash rack for several hours. Certainly the low velocities in the intake section were the reason for this. No fish was forced to the trash rack and they could freely swim in the upstream basin which had an area of about 30 times the intake cross section. In the following the analysis of the results remains restricted to the ensemble of all five fish species. As only 38 fish were killed the ensemble for analyses gets very small if sub-ensembles like differing species were considered. Only the distinction between differing intake velocities had been made and the two extrema, namely 0.5 m/s and 0.3 m/s intake velocity have been further analyzed. For the analysis all fish in the various tests had been pooled in clusters of approximately equal length. Then for each size cluster averages were computed and a best fitting curve through the weighted averages were computed, considering the number of individuals in each average as weight factor. This has been done first of all for the size dependent probability of bypass passage (Figure 4), subsequently for the mortality in the turbine considering two differing intake velocities (Figure 5) and finally for the combined size dependent damage probability as a product of the two previously described probabilities (Figure 6). A very interesting and significant observation could be made related to the size-dependent probabilities whether to use the provided bypass migration path or the dangerous migration path through trash rack and turbine. On the one hand side the probabilities of smaller fish are higher to be attracted by the flow to the

Figure 5. Damage rate in turbine depending on fish lengths

and intake velocity. Diameters of size-class average indicate

the individual numbers in each size-class.

turbine. This effect is nicely shown in Figure 4. One

can clearly see, that a power function for fish length

and probability of bypass passage very well describes

the observations. Also there is a clear influence of the

intake velocity on the bypass use. On the other hand, the probability of damage by the

turbine is higher for longer fish. From literature (e.g.

Montén, 1985) one can expect a linear relation between

fish length and mortality. This is shown in Figure 5.

Even though the ensemble for statistical analysis is small due to the few individuals killed a trend on the differing intake velocities can be observed. There is a higher mortality for 0.3 m/s intake velocity than for 0.5 m/s intake velocity. This has to do with the fact, that the intake velocity is regulated with the turbine. The turbine therefore was throttled and not completely opened with blades and guide vanes for the smaller velocity. The interesting thing is now that the effects of bypass probability and turbine mortality partly com pensate. While a longer fish has a low probability of choosing the migration path through the turbine, it has, for this case, a high probability of lethal damage in the turbine. Therefor the two effects partly compensate. Figure 6 shows the combination of the two probabil ities and illustrates that a maximum, size dependent damage results for a medium fish length, whereas for shorter and longer fish the probability drops to zero. If one considers the integral of the mortality curves and computes an average mortality rate over the entire range of endangered fish length, i.e. from 0 mm up to about 20 mm the following average mortalities can be computed: For 0.3 m/s intake velocity the average is 4.5% and for the higher velocity of 0.5 m/s it amounts to 7.2%. If one now tries to upscale the above figures to real and therefore larger powerplants the injury and mor

tality rate would decrease due to the more favorable

geometric and operational conditions, i.e. the increase

of diameter and decrease of turbine RPM, by at least

a factor of 2 (400 kW powerplant with 2 turbines). It should be stressed that all graphics related to

injury or mortality rates and all the corresponding fig

Ebel G. (2013). Fischschutz und Fischabstieg anWasserkraftanlagen – Handbuch Rechenund Bypasssysteme. Ingenieurbiologische Grundlagen, Modellierung und Prognose, Bemessung und Gestaltung, Mitteilungen aus dem Büro für Gewässerökologie und Fischereibiologie, Band 4, Dr. Ebel, Halle (Saale), Germany.

Geiger F., Rutschmann P. (2014). Experimental Investigation of Fish Protection and Fish Downstream Passage for Small Fish at the TUM-Hydro Shaft Power Plant, Proceedings of the 10th International Symposium on Ecohydraulics,

Trondheim. Geiger F. Schäfer S., Rutschmann P. (2015). Fish damage and fish protection at hydro power plants. Experimental investigation of small fish under laboratory conditions, EProceedings of the 36th IAHR World Congress, 28 June – 3 July 2015, The Hague, the Netherlands, 6 pages. Montén E. (1985). Fish and Turbine – Fish Injuries During Passage Through Power Station Turbines, Vattenfall, Stockholm, ISBN 91-7186-247-1. Rutschmann P., Sepp A. (2011). A new take on hydro power design, International Water Power & Dam Construction, Vol. 63, No. 3, pp. 22 – 26. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Long-term evaluation of the wave climate and energy potential

in the Mediterranean Sea

George Lavidas

Institute for Energy Systems, University of Edinburgh, Scotland, UK,

CERES Global, European Offices, Brussels, Belgium

Vengatesan Venugopal

Institute for Energy Systems, University of Edinburgh, Scotland, UK

Atul Agarwal

CERES Global, Asia offices, Bangalore, India

ABSTRACT: Waves are characterised by high energy density content, able to provide significant renewable

energy contribution. Identification of wave energy potential locations around Europe can provide significant

benefits for application of wave energy converters (WECs). While the highest resources are placed in the open

Atlantic coastlines of Europe that does not exclude the fact that wave energy can provide significant contributions

to the rest of Europe. To support such claims, a detail resource assessment has to be in place to characterize the resource. State

of-the-art resource assessments provide a tool that can identify trends in the resource and proper evaluation of

production, based on such long-term data allows robust financial and engineering decisions. In addition, the

results can also be used for climate investigations, extreme value analysis, site characterization, and information

concerning a variety of offshore activities. This study investigates the metocean conditions of the Mediterranean Sea and assesses the levels of energy

around the Basin. Hidden opportunities lay in both energetic and milder wave environments with each having

each own benefits and drawbacks, in terms of capital costs required and energy production. The results from the

hindcast underwent a detailed annual, seasonal, and decadal analysis. Furthermore, potential benefits of wave

energy converters (WECs), are assessed in terms of energy

production, adaptability, and final considerations for site selection are discussed.

1 INTRODUCTION

Energy plays a significant role in the technological and financial growth of every country. While energy requirements can be met by numerous sources i.e. nuclear, fossil, renewable, one key component is to ensure energy security and stability. The option of renewable resources offers abundant and significant opportunities to increase energy independence and offer emission free electrical production. In order to achieve the targets for emissions reduc

tion, increased renewable energy production, as these are adapted into the European Policy Framework (Par

liament 2009) all renewable resources have to be accounted for and develop plans for utilization.Adding various renewable technologies to the energy mix, will complicate the production patterns, but will reduce the intermittent production significantly. Leading to higher degrees of penetration by renewables, reduc

ing associated infrastructure expenditures and energy imports (Delucchi and Jacobson 2011, Jacobson and Delucchi 2011, Schaber et al. 2012). Waves offer a renewable resource that is abundant and presents much higher energy density than its direct contributor wind, due to the nature of the fluid (Barstow et al. 2009, Clément et al. 2002, Gunn and Stock-Williams 2012, Reguero et al. 2015). The exploitation of waves as an energy source has been suggested by many studies and the evolution of wave energy converters (WECs) throughout the years has added information to unlocking the potential for energy production (Clément et al. 2002, Cruz 2008, Falcão 2010). Although the benefits from ultimately utilizing waves as energy production are numerous, there are quite a lot of uncertainty affecting the proper estimation of WECs production, regional adaptability, and survivability. Early work in the wave energy field, identified as most energetic waters the regions exposed to the mid and northAtlantic ofWestern coastal Europe

(Barstow et al. 2009, Pontes et al. 1996). Resource assessments for wave energy have evolved using atmo spheric, wave numerical models, which allow us to investigate through hindcasts (historical), and fore cast studies. The application of numerical modelling to generate long-term data has been outlined by var ious studies and has given confidence in their usage, if the model is properly constructed, and can provide necessary metocean data (Caires and Sterl 2005, Sterl and Caires 2005, Mackay et al. 2010a, Mackay et al. 2010b, Vinoth andYoung 2011). The construction of a numerical model though is a not an easy process with, many factors affecting the final outcome and several considerations taken into account when adapting to a region (van Vledder et al. 2010, Zijlema et al. 2012, Reguero et al. 2015).

2 MODEL CONSTRUCTION

Our study focuses on the Mediterranean region using a third generation wave numerical model, Simulating WAves Nearshore (SWAN). SWAN is a third gener ation phased-averaged highly skilled model, that has the ability to solve the wave kinematic equation in Cartesian or Spherical coordinates. It can take into account nearshore complex parameters that are not accounted for in larger models, diffraction, non-linear triad interactions, shoaling, and coastal bottom break ing.The main driver for the generation and propagation of waves is the designated wind input used by the user (Delft 2014). The SWAN version used in this study is 41.01A, activated with Spherical coordinates and using a two-way nested scheme approach.The primary mesh used is 0.1° while four additional smaller domains are also utilised covering the majority of the countries exposed by the Mediterranean Sea. The subsequent meshes have a spatial resolution of 0.025°, these higher resolution domains enhance the computations concerning the coastline and non-linear interactions providing long-term coastal data, see Figure 1.

As driving winds the CFSR Re-Analysis dataset (Saha et al. 2010) was chosen, since previous expe rience showed that the temporal resolution of the product reduces the “missing” peaks, that is a com mon phenomenon in wave numerical models (Cavaleri 2009).The authors have previously tested several other products for the region with some of the work men tioned (Lavidas et al. 2014b) and work currently in development. For both domains (coarse and fine) the same wind product was used. The direction was subdivided 15 ∘ intervals while the frequency resolu tion has a low limit 0.035 and highest 0.5 (from 2 seconds-28 seconds) and thirty increments distributed logarithmically.

The initial and subsequent nested spectral infor mation based on a JONSWAP spectrum with a peak enhancement factor for wind-generated seas, bottom friction, breaking, refraction, diffraction, and triad interactions activated. Finally, the non-linear quadru plet interactions are using a semi-implicit solution. Figure 1. Bathymetry in meters, the nested areas utilize a much higher resolution. Figure 2. Differences in meters for H sig . The orography of the region is characterised by rapid changes in the bathymetry, significant amount of island and complex coastlines that increase non-linear consideration, thus to ensure numerical performance a backwards step and time propagation is used. The selection of the wind also prompted us to compare the performance of the model under different schemes of generation, since the high temporal resolution (1-hour) affects the wave resource hindcasted. For this reason we tested the adaptation and combination of customized solutions with a linear wind growth coefficient and two wind generation options based on the work of (Komen et al. 1994) now denoted WAM3, and (Janssen 1988) denoted as WAM4. The difference of the schemes lay on the way that the wind drag coefficient is estimated, based on the wind stress. The difference in the final metocean products based on the two wind schemes showed that the WAM3 offers better approximation with the buoy recordings, and thus was selected as the most appropriate to use throughout the hindcast. While both of them showed a high correlation in terms of generation, the combination of high-resolution data and the re-computed wind drag offers over-estimation if the second scheme is used, see Figure 2. For validation buoys available by the (Hellenic Centre for Marine for Research 2014) network are take into account as we calibrated and then validate the model, while some level of post-processing was applied to

Table 1. Validation for the models and buoys available recordings

Indices	Athos (2000–2014)			Lesvos (2000, 2002–2012)			Mykonos (2002–2012)			Pylos (2007–2014)		
	H sig	T peak	T m02	H sig	T peak	T m02	H sig	T peak	T m02	H sig	T peak	T m02
R	0.95	0.87	0.92	0.93	0.86	0.92	0.87	0.67	0.77	0.93	0.91	0.93
RMSE	0.34	1.11	0.74	0.39	1.05	0.64	0.52	1.68	0.87	0.38	1.06	0.73
MPI	0.98	0.90	0.92	0.98	0.90	0.93	0.98	0.89	0.92	0.97	0.85	0.89
Av. Buoy	0.81	4.56	3.66	0.76	4.57	3.53	1.00	4.82	3.63	0.98	5.83	4.36
Av. SWAN	0.82	4.45	3.24	0.89	4.45	3.25	0.87	4.70	3.26	0.99	5.59	3.96
Bias	0.01	−0.11	−0.42	0.13	−0.12	−0.28	−0.13	−0.12	−0.37	0.01	−0.24	−0.40
SI	0.41	0.24	0.20	0.52	0.23	0.18	0.52	0.35	0.24	0.39	0.18	0.17

the raw data of the buoys since they contained significant and distinct levels of "noise spikes" and missing intervals. The hindcast of the metocean conditions was performed for a 35 years period from 1980 to the end of 2014, the availability of the buoys although does not span for the same amount of time, since the POSEIDON system is active since 2000 (Papadopoulos et al. 2002). The validation results are in good agreement with a recent study (Zacharioudaki et al. 2015) that used the WAM model to extract a hindcast for a long term period for the Greek region. Several buoys are available although due to length considerations, overall results of the validation are presented in Table 1. The locations taken into account are dispersed around the Aegean Sea, Athos located at the North, Lesvos at North East, Mykonos Central Aegean, and Pylos at the SouthWest of the Ionian Sea.The selection of the buoys considered present the ability of the model to various sea-state as validated by the buoys. Majority of the differences is located at the biases, Athos and Pylos being close to zero, while Lesvos and Mykonos show an over and under-estimation respectively. The first two locations, are deep-water locations (Athos, Pylos) with no major coastal (land masses) surrounding them. Lesvos is located just offshore the island of Lesvos and exposed to a low resource, Mykonos on the hand represents one of the most challenging buoy locations to hindcast. It is located within the Cycladic complex, decreasing the propa gated resources by the presence of turning coastlines and island complexes. The sudden depth variations also pose a significant issue especially for coastal mod elling, implying that for an even more detailed analysis of a small area, the use of a third nest would be desir able with the potential use of unstructured meshes.The customized model offers improvements from previous work undertaken in the area by the authors with biases

reduced significantly by 30% (Lavidas et al. 2014a).

3 WAVE ENERGY

The results from the hindcast were adapted to estab

lish a database with the energy potential around

the Mediterranean Sea, similar attempts have been undertaken by previous studies with most recent one (Medatlas Group 2004, Ratsimandresy et al. 2008, Zacharioudaki et al. 2015) focusing on wave height and (Soukissian et al. 2008, Ayat 2013, Liberti et al. 2013, Vicinanza et al. 2013, Monteforte et al. 2015) focusing on wave energy. Both categories do complement each other, although in all cases the models used focused predominately in specific regions, used coarser model resolutions not resolving adequately coastal environments and/or had limited time duration, commonly 10 years. In term of temporal duration, exceptions are a 44 year hindcast (Ratsimandresy et al. 2008) focusing on Spanish coasts for 42 years and (Zacharioudaki et al. 2015) focusing in the Aegean sea for 42 years, both using the oceanic model WAM. In terms of spatial resolution (Liberti et al. 2013) used a model appropriate for coastal representation at resolution 0.060 ∘ degrees, while of the majority of studies are conducted with spatial information between 1 ∘ and 0.5 ∘ degrees. Our nested attempt allows us to enrich the knowledge we have on the wave conditions and power density around the region, and subsequently its nested meshes. Having established the validity and ability of the hindcast additional location where extracted, focusing on coastal areas and low depths for which WECs present immediate interest of installation. The overall estimates of annual and seasonal wave energy contents are shown in Figures 3-4. 3.1 Wave energy resource Most energetic waters can be located at the coastal North-East Spanish coasts, South and North-East Italian coasts, Central Greece island complexes, North Tunisia and the North coastlines of Africa (LibyaEgypt). Both the annual and seasonal alterations exhibit a distribution of wave power dominated by wind-wave seas and mid-long fetches. The presence of many islands, the rapid bathymetric changes, and coastlines decrease the annual amount of wave power in values ranging from 8–16 kW/m.The seasonal analysis shows though that at specific region wave power (seasonally) can reach values up to 20–25 kW/m. WECs application has interest in depths of up to 150 m, due to the orography of the sea-bed sharp

Figure 3. Annual Wave Power distribution for for the hindcast dataset.

Figure 4. Seasonal distribution of wave power (kW/m) over the hindcast period.

changes are present posing a barrier on some areas,

while this also acts as a natural obstacle for full wave

evolution. With the ability of the model to repre

sent coastal locations, through use of high-resolution

bathymetry, several locations (over 80 through the

nested runs not shown here), have been extracted

and their hourly distributions have been separated in

months, decades and overall resource levels, allowing

a better understanding of potential opportunities.

The results show that the resource is Italy is more

constant in terms of magnitude between 6–15 kW/m

taking into account the location of investigation South

West or East at the Adriatic Sea. In the Greek Aegean

region, the wave power levels are mostly located in cen

tral and South Aegean, with values from 7–12 kW/m

at non-complex location (i.e. not sheltered between the many islands) and in the Northern part form 4–7 kW/m. The Southern coastline of France between Montpellier and Perpignan, due to their enclosed status, non-linear-shallow water interactions, diffraction, and shoaling effects exhibit lower values ranging from 3–6 kW/m, while on the South East of France Cote d Azur the values have a slight increase from 4–7 kW/m. At the Spanish coastlines, more specifically the Balearic Sea locations around the island of Mallorca have values from 8–15 kW/m and the coastal location closer to continental Europe are lower around 7-10 kW/m. In the exposed Libyan coastlines average values range from 6-11 kW/m with most of the energy located at the North-West Libya. Finally, Tunisia presents also a variation of energy content, with the Northern coastlines having higher resource

Figure 5. Covariance of wave energy levels around the basin

(reduced scale).

of up to 12 kW/m, while the East side with inwards

turning coastlines reduce significantly to 2–5 kW/m.

3.2 Variability of the resource

Apart from the level of available resource, one has

to consider that the annual levels of variation present

an important factor that needs to be considered when

exploring optimal wave energy sites. As discussed the

levels of wave energy in the Mediterranean region

are almost three times less than open Atlantic coasts,

although their annual variation is significantly lower.

With that in mind the expected production of WECs

will be increased in terms of temporal operation, while

the inherit problems associated with higher resources

are also minimized i.e. extreme waves, fatigue on

WECs, survivability etc. The covariance of wave energy aids in the determi

nation of locations with satisfying energy content, and

minimal annual distortions in the wave energy flux. As

expected the covariance is increased at coastal areas

and complex terrains, see Figure 5, nevertheless the

recorded levels of expected variation are significantly less of those met in open seas. This poses the opportu nity for operation of properly selected WECs, ensuring stability in operation, production, and enhancing the level of reliability for wave energy. In Figure 5 the covariance of the energy distribu tion is higher at Northern parts of Greece, implying annual variations that may disrupt production. The majority of coastlines exposed to longer fetches like the South Italian coasts, Spain,Tunisia, Libya and Cen tral Greece record low levels of variation. The nested approached used high-resolution meshes (not shown here) allowed verification and further improvements in the perception of this variation in coastal areas. In Greece, higher levels are found in the North ern coastlines of Thrace and Macedonia, followed by high levels at then Straits of Euboia. The identified high energy potential of the region at Central Aegean, presents low variations, especially in the high energetic region of the Cyclades Island complex. For France the coastlines along the French Riviera have a low vari ation often less than 0.1, while the sheltered location reach values of 0.3–0.4. In Spain throughout the South coastlines, the covariance is low with high levels of available seasonal resource, while the coastlines around Mallorca present higher levels of variation. For Italy, Sicily has low levels 0.09–0.15, while the East part exposed to the Ionian and Adriatic seas have a higher level around 0.25. In Libya with the exception of the Gulf of Sidra (KhalŁ J Surt) with variation from 0.35–0.4, rest of the coastlines so little or no variation. Finally, Tunisia presents higher variation at the peninsula of Jerba at the

Gulf of Gabes, with values around 0.25. Table 2 presents the estimated mean wave power levels (kW/m) over the hindcast period, the standard deviation and percentile recorded. Due to the length of the hindcast, these results corresponds to the entire period. Similar analysis has been done to the locations for annual, seasonal and monthly intervals. The results present small annual standard deviations, and lower annual percentiles (from year to year). The authors have produced over 80 locations with their corresponding wave information, although not are able to be presented here. The energy analysis in combination with the variance of the annual and overall 35 year resource, can aid to selection of energetic sites with consistent exposure. Such considerations are expected to fully utilise low frequency energy converters and deliver a consistent flow of energy production by WECs, with extreme events and structural fatigue to be significantly less. 4 CONCLUSIONS A high-resolution third generation numerical model (SWAN) was applied to the region of the Mediterranean to produce a long-term database of wave energy that can be used in the application of wave energy converters. The considerations on selecting the wind dataset, calibration and validation where presented, confirming the reliability of the model. The results present the wave energy distribution both annual and seasonal, around the Basin and the potential available power resource in many countries. Based on the long duration of the study, and the use of a customised coastal model, wave energy quantification is enhanced for not only deep water locations but more importantly for coastal sites, where WECs can be deployed. While some literature and data exist for the region, mostly oceanic models have been used, which are not suitable for coastal quantifications. Following the wave energy content examination, a focus is given on the covariance and variability the resource. This allows to draw conclusions concerning the expected variation of wave energy production by WECs’. The variation of the resource is dependent on the local characteristics of every area investigated, with coastal and island complexes (i.e Greek coastlines) presenting the higher levels. Italy and Spain present the highest potential with low covariance, while for the first time the quantification of energy content of Tunisia and Libya are presented.

Table 2. Mean Wave Power content (kW/m) over the whole datasets, standard deviation and percentiles encountered

Greece	Athos	Attika	Crete1	Crete2	E1mea	Euboia	Kythnos	Lesvos	Mykonos	Naxos
Power	3.89	1.71	3.54	5.28	2.87	2.96	3.68	4.06	2.87	2.44

std (σ) 10.34 3.82 8.44 10.59 7.71 9.10 7.94 9.64 4.61 4.32

95 th Perc. 18.99 7.69 15.29 21.12 12.39 13.93 16.51 18.87 10.75 10.10 Greece (cont.) Tunisia Libya Paros Pylos Santorini Tunisia1 Tunisia2 Libya1 Libya2 Libya3 Libya4 Libya5

Power 2.89 4.74 4.64 6.16 1.69 3.94 4.58 3.24 3.24 3.38

std (σ) 4.85 11.06 8.75 13.30 3.46 8.75 10.84 6.06 6.06 6.98

95 th Perc. 11.70 21.06 19.03 27.00 6.85 15.47 19.80 11.71 11.71 12.95 Italy Desil GasilA GasilB Italy1 Italy2 Italy3 Italy4 Italy5 Italy Italy7

Power 6.86 4.95 6.06 4.10 4.13 2.50 3.26 1.78 2.55 2.24

std (σ) 15.36 10.37 12.41 9.04 10.54 5.42 6.74 5.99 8.16 6.40

95 th Perc. 30.24 20.72 25.38 18.08 19.31 10.71 13.50 8.10 11.32 10.03 Italy(cont.) Spain Mazzaro Palermo RonMazzaro Tauro Alghero Barcelona1 Barcelona2 Capder Palamos

Power 5.12 5.12 5.12 1.97 8.03 1.27 1.24 3.76 2.09

std (σ) 11.46 11.46 11.46 6.90 19.7 3.15 2.99 8.61 5.01

95 th Perc. 22.53 22.53 22.53 8.61 39.8 5.50 5.35 16.13 8.93 Spain(cont.) France Spain1 Spain2 Spain3 Spain4 Fr61191 Fr61284 Fr61289 Fr6190 France1 France2

Power 0.93 1.28 4.50 7.13 1.21 1.81 2.43 1.16 1.17 1.95

std (σ) 2.45 3.81 11.14 18.35 4.52 4.19 4.62 4.19 4.30 4.57

95 th Perc. 3.59 5.37 20.14 34.35 4.49 7.18 10.16 4.93 4.73 7.75

While the levels of available energy are not as high as oceanic coastal locations, the variation and expo sure to extreme events are reduced, allowing the further consideration for wave energy applications as viable solutions. The low levels of “peak” events may prove beneficial for WECs considering that this ensures reli

able operation and lead to faster proof-of-concept;

while simultaneously will promote the adaptability

and verification of WECs operation.

The long coastlines enclosed in the Mediterranean

countries can provide a significant resource for

the future production of renewable energy, adding

wave energy as a candidate for consideration in the

de-carbonisation and increased energy security of any

country.

ACKNOWLEDGEMENTS

The authors would to express their gratitude for the

buoy access, wind, and bathymetry provided by sev

Delft, T. (2014). SWAN scientific documentation Cycle III version 41.01. Delft University of Technology Faculty of Civil Engineering and Geosciences Environmental Fluid Mechanics Section.

Delucchi, M. a. & M. Z. Jacobson (2011, mar). Providing all global energy with wind, water, and solar power, Part II: Reliability, system and transmission costs, and policies. Energy Policy 39(3), 1170–1190.

Falcão, A. F. D. O. (2010, apr). Wave energy utilization: A review of the technologies. Renew. Sustain. Energy Rev. 14(3), 899–918.

Gunn, K. & C. Stock-Williams (2012). Quantifying the global wave power resource. Renew. Energy 44, 296–304.

Hellenic Centre for Marine for Research, H. (2014). Monitoring, Forecasting System Oceanographic information for the Greek Seas (POSEIDON).

Jacobson, M. Z. & M. a. Delucchi (2011). Providing all global energy with wind, water, and solar power, Part I: Technologies, energy resources, quantities and areas of infrastructure, and materials. Energy Policy 39(3), 1154–1169.

Janssen, P. A. (1988). Wave induced Stress and drag of air flow over sea waves.pdf. J. Phys. Oceanogr. 19(6), 745–754.

Komen, G., L. Cavaleri, M. Donelan, S. Hasselmann, & P. Janssen (1994). Dynamics and Modelling of Ocean waves. Cambridge University Press.

Lavidas, G.,V.Venugopal, & D. Friedrich (2014a). Investigating the opportunities for wave energy in the Aegean Sea. In 7th Int. Sci. Conf. Energy Clim. Chang. 8-10 Oct. 2014 Athens, Athens. PROMITHEAS The Energy and Climate Change Policy Network.

Lavidas, G., V. Venugopal, & D. Friedrich (2014b). On Investigating Wind -Wave Resource To Enhance Predictability In Offshore Wave Energy Deployments. In Int. Conf. Offshore Renew. Energy 2014 ASRANET 15th-17th Sept. Glas. 2014, Glasgow.

Liberti, L.,A. Carillo, & G. Sannino (2013, feb). Wave energy resource assessment in the Mediterranean, the Italian perspective. Renew. Energy 50, 938–949.

Mackay, E. B.,A. S. Bahaj, & P. G. Challenor (2010a). Uncertainty in wave energy resource assessment. Part 1: Historic data. Renew. Energy 35(8), 1792–1808.

Mackay, E. B., A. S. Bahaj, & P. G. Challenor (2010b). Uncertainty in wave energy resource assessment. Part 2: Variability and predictability. Renew. Energy 35(8), 1809–1819.

Medatlas Group (2004). Wind and Wave Atlas of the Mediterranean Sea. Technical Report April, Western European Union.

Monteforte, M., C. Lo Re, & G. Ferreri (2015). Wave energy assessment in Sicily (Italy). Renew. Energy 78, 276–287.

Papadopoulos, A., P. Katsafados, G. Kallos, & S. Nickovic (2002). The Weather Forecasting System for Poseidon – an Overview. J. Atmos. Ocean Sci. 8(2-3), 219–237.

Parliament, E. (2009). The European Parliament. Directive of the European Parliament and of the Council on the promotion of the use of energy from renewable sources amending and subsequently repealing directives 2001/77/ec and 2003/30/ec. Technical report. Pontes, M. T., G. a. Athanassoulis, S. Barstow, L. Cavaleri, B. Holmes, D. Mollison, & H. Oliveira-Pires (1996). An Atlas of the Wave-Energy Resource in Europe. J. Offshore Mech. Arct. Eng. 118(4), 307. Ratsimandresy, a. W., M. G. Sotillo, J. C. Carretero Albiach, E. Álvarez Fanjul, & H. Hajji (2008). A 44year high-resolution ocean and atmospheric hindcast for the Mediterranean Basin developed within the HIPOCAS Project. Coast. Eng. 55(11), 827–842. Reguero, B., I. Losada, & F. Méndez (2015). A global wave power resource and its seasonal, interannual and long-term variability. Appl. Energy 148, 366–380. Saha, S., S. Moorthi, H. L. Pan, X. Wu, J. Wang, S. Nadiga, P. Tripp, R. Kistler, J. Woollen, D. Behringer, H. Liu, D. Stokes, R. Grumbine, G. Gayno, J. Wang, Y. T. Hou, H. Y. Chuang, H. M. H. Juang, J. Sela, M. Iredell, R. Treadon, D. Kleist, P. Van Delst, D. Keyser, J. Derber, M. Ek, J. Meng, H. Wei, R. Yang, S. Lord, H. Van Den Dool, A. Kumar, W. Wang, C. Long, M. Chelliah, Y. Xue, B. Huang, J. K. Schemm, W. Ebisuzaki, R. Lin, P. Xie, M. Chen, S. Zhou, W. Higgins, C. Z. Zou, Q. Liu, Y. Chen, Y. Han, L. Cucurull, R. W. Reynolds, G. Rutledge, & M. Goldberg (2010). The NCEP climate forecast system reanalysis. Bull. Am. Meteorol. Soc. 91(8), 1015–1057. Schaber, K., F. Steinke, P. Mühlich, & T. Hamacher (2012, mar). Parametric study of variable renewable energy integration in Europe: Advantages and costs of transmission grid extensions. Energy Policy 42, 498–508. Soukissian, T. H., A. Prospathopoulos, G. Korres, A. Papadopoulos, H. Maria, & K. Maria (2008). A new wind and wave atlas of the Hellenic Seas. In Proc. ASME 2008 27th Int. Conf. Ocean. Arct. Eng. OMAE2008, Estoril,Portugal, pp. 1–9. ASME. Sterl, A. & S. Caires (2005). Climatology, variability and extrema of ocean waves: The web-based KNMI/ERA-40 wave atlas. Int. J. Climatol. 25(7), 963–977. van Vledder, G., M. Zijlema, & L. Holthuijsen (2010). Revisiting the JONSWAP bottom friction formulation. In Proc. 32nd Conf. Coast. Eng. Shanghai, China, 2010, pp. 1–8. Proceedings of the International Conference on Coastal Engineering; No 32. Vicinanza, D., P. Contestabile, & V. Ferrante (2013). Wave energy potential in the north-west of Sardinia (Italy). Renew. Energy 50, 506–521. Vinoth, J. & I. R. Young (2011, mar). Global Estimates of Extreme Wind Speed and Wave Height. J. Clim. 24(6), 1647–1665. Zacharioudaki, A., G. Korres, & L. Perivoliotis (2015). Wave climate of the Hellenic Seas obtained from a wave hindcast for the period 19602001. Ocean Dyn. 65(6), 795–816. Zijlema, M., G. van Vledder, & L. Holthuijsen (2012, jul). Bottom friction and wind drag for wave models. Coast. Eng. 65, 19–26. Sustainable

Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Numerical simulation and validation of CECO wave energy converter

P. Rosa-Santos, M. López-Gallego & F. Taveira-Pinto

Interdisciplinary Centre of Marine and Environmental Research (CIIMAR) and Faculty of Engineering of the

University of Porto, Department of Civil Engineering, Porto, Portugal

D. Perdigão

Faculty of Engineering of the University of Porto, Porto, Portugal

J. Pinho-Ribeiro

PT Portugal, Porto, Portugal

ABSTRACT: CECO is a new wave energy converter (WEC), designed to convert simultaneously the kinetic and the potential energy of waves in electricity, based on the oblique motion of two lateral floating modules. Its proof of concept was successfully carried out at the Hydraulics Laboratory of the Hydraulics, Water Resources and Environment Division of the Faculty of Engineering of the University of Porto, Portugal, using a physical model built on a geometric scale of 1/20. The results obtained are used in this paper to validate a hydrodynamic numerical model of CECO created withAnsys ® Aqwa™, which is a code based on the boundary element method. The experimental proof of concept and its main conclusions are briefly described, and the subsequent numerical validation work is presented. Initially the results of the physical model tests are used to calibrate the numerical model, to correctly simulate the mechanical losses and damping in the system. Once calibrated, the model is used

to analyze the behavior of CECO under new wave conditions, to better understand its behavior and performance.

The numerical results showed a good agreement with the physical model ones and allowed to obtain a better

insight into the performance of CECO. The efficiency in capturing the wave energy was estimated to exceed

50% for some wave conditions. Although the results obtained are very promising, additional research is required

to fully optimize CECO performance.

1 INTRODUCTION

Ocean waves are an important renewable energy resource with several recognized advantages: pre dictable characteristics (1 to 2 days in advance) with respect to other renewable energy resources, limited ecological and environmental impacts and different timing of intermittency if compared, e.g., to wind and solar energy (Taveira-Pinto et al., 2015). If intensively exploited, wave energy might significantly contribute to the energetic mix of many ocean facing countries, reducing the need of non-renewable sources and their drawbacks (López et al., 2015).

A significant number of technologies to transform wave energy into electricity was developed over the last decades, and some of the more promising concepts are undergoing demonstration testing at (near) full scale (Kofoed et al., 2006; Elwood et al., 2010). In the development process of wave energy converters, from concept validation up to the multi-device demonstra

tion in real ocean conditions, several stages have to be

fulfilled in order to reach reliable, optimized and cost

effective solutions and minimize investment risks. In

that five staged process (Heller, 2012), physical and numerical modelling applied in a complementary way are needed. This work aims to present the progress in developing the CECO wave energy converter achieved at the Faculty of Engineering of the University of Porto, Portugal.The main advances and conclusions obtained through physical modelling during the last years (Marinheiro et al., 2015; Rosa-Santos et al., 2015a) and the more recent numerical simulation work using Ansys ® Aqwa™ are summarized (Ansys, 2013).The latter was calibrated with the results of the physical model tests and, once validated, was used to analyze the behavior of CECO under new wave conditions for two configurations: with and without the PTO system. On the basis of the numerical results obtained, the ability of CECO to capture wave energy and to transmit it to the PTO is shown and discussed. This paper is structured as follows. Section 2 presents CECO general concept and, in the subsequent Section 3, the proof of concept studies carried out are summarized along with the main results gathered. The numerical modeling of the WEC is presented in Section 4, including the model description, the calibration and the analyses of the results. Finally, the main

Figure 1. Components of CECO: (1) central element, (2)

lateral floating modules, (3) supporting frame and longitudi

nal rods, and (4) direction of movement.

conclusions obtained during the development of

CECO are presented in Section 5.

2 CECO WAVE ENERGY CONVERTER

CECO wave energy converter has two lateral float

ing modules that, under the action of waves, have an

oblique motion in relation to a fixed central cylindrical element (Marinheiro et al., 2015; Rosa-Santos et al.,

2015a), Figure 1. The inclination and direction of CECO's motions

can be modified according to the characteristics of

incoming waves, in order to ensure the best possible

performance of this one degree of freedom device.This

is done by changing the inclination of the supporting

elements and by aligning the device with the direction

of incident waves, Figure 2. This concept was idealized as to allow the simul

taneous absorption of the kinetic and the potential

energy of waves, not favoring one of those compo

nents in particular. It is therefore expected that the

device would be able to absorb a significant part of

the energy transported by ocean waves (Marinheiro

et al., 2015; Rosa-Santos et al., 2015b). It is difficult to include CECO into any of the typ

ical categories of WEC (Falcão, 2010). On the one

hand, the floating modules must have an alignment

parallel to wave fronts, but the total width of CECO is

not sufficiently large to define it as a terminator. On

the other hand, as in point absorbers, CECO dimen

sions are quite small in comparison with typical wave

lengths. However, most point absorbers (e.g., heaving

type axisymmetric) are able to naturally absorb wave

energy from any direction while CECO has to be Figure 2. Modification of CECO inclination (up) and orientation (down). properly orientated to waves to avoid being negatively affected in terms of performance. In addition, most point absorbers operate only in the heaving mode (Richardson and Aggidis, 2013) and are not able to harvest energy in the other oscillation modes. Since CECO moves in an oblique direction, it is expected to efficiently absorb wave energy in two directions (surge and heave). It is also important to mention that point absorbers are usually designed to resonate for typical wave frequencies and present a narrow frequency response curve with a large peak. This issue can be overcome with the incorporation of tuning mechanisms that increase the complexity and overall cost of the device. The work already carried out has shown that under certain conditions, CECO has a flatter frequency response curve, presenting good efficiencies for a relatively large range of sea conditions. 3 PROOF OF CONCEPT STUDIES CECO wave energy converter is being developed at the Hydraulics Laboratory, of the Hydraulics, Water Resources and Environment Division of the Faculty of Engineering of the University of Porto, Portugal, since 2012. Two physical model studies were carried out on a multidirectional wave basin that is 28.0 m long, 12.0 m wide and 1.2 m deep. The tested physical models were scaled-down reproductions of the idealized CECO prototype built on a geometric scale of 1:20. The Froude similitude criterion was adopted in the studies. The objective of the first study was to validate the concept and working principle of CECO, and to analyze the feasibility of some initial constructive solutions (Teixeira, 2012). In the second study, based

on the knowledge acquired in the preceding one, sev

eral improvements were introduced in the physical

model, mostly in the geometry of the floating modules,

the guiding system of the main rods, the cross-section

of the central element, as well as in the simulation and

monitoring of the power-take-off (PTO) system, which

started to allow the direct measurement of the instan

taneous power absorbed by the device (Marinheiro,

2013).

The experimental equipment used consisted of resistive wave gauges to measure the water free sur face elevation (accuracy of ±0.4 mm), the system to measure the power produced by the PTO, and a motion capture system composed of two digital infrared cam eras, able of measuring the motions of floating bodies, in their six degrees of freedom (surge, sway, heave, roll, pitch and yaw), without contact with the model (accuracy of ±0.5 mm for translational motions and ±0.1 ∘ for rotational motions). A sampling frequency of 40 Hz was selected to avoid aliasing in data spec tral analysis and to provide a satisfactory accuracy to the parameters estimated based on time domain analysis. In the first experimental study, using a simplified

geometry and PTO system, it was concluded that the response of CECO occurs, mainly, in the frequency of incident waves during the entire test duration and that its performance strongly depends on the incident wave characteristics and the damping introduced by the PTO. The relative capture width, which represents the ratio between the mean absorbed power (i.e. the mean value of the absorbed power time series) and the power contained in a wave front with the width of the device, reached 14% showing that CECO is a valid technology to harvest wave energy (Rosa-Santos et al., 2015a). The calculated absorbed power did not take yet into account the losses related to the conver sion of mechanical energy into electricity; however, it is important to take into account that a simplified physical model of CECO and a not yet optimized PTO were used. The inclination angle of 45 ∘ (relative to the horizontal) was the most favorable one for the tested geometry and wave conditions. The system used to guide the lateral floating modules was found to be a key component of this technology, which is not surpris ing since this WEC is based on the relative motions between a floating mobile parte and a fixed central element. In addition, it was found that in the tests with reg ular waves, CECO followed faithfully the water free surface elevation. However, in the tests with irregu lar waves, when waves with too short wave periods or wave heights reached the device during the floating modules downward motion, the upward motion could be jeopardized, since there was no time or momentum to decelerate the downward motion and reverse direc tion. This type of response showed the importance of the mass, dimensions, shape and submergence level of CECO floating modules. It was then stated that these subjects should be analyzed in more detail in the future using numerical modelling. In the second phase, with an improved and more complex model, which allowed the direct measurement of the electrical potential differences produced by the PTO and, therefore, taking into account the losses related to the process of converting wave energy into electricity, relative capture widths between 10 and 30% were obtained for the relevant test conditions (Marinheiro et al., 2015; Rosa-Santos et al., 2015b). It was also mentioned that the tested CECO geometry and PTO system seemed to be more favorable for the shortest wave period (8 s), with the efficiency decreasing with the wave period. It was again concluded that the inclination angle was an important factor controlling the behavior of CECO. For all tested conditions, results were better for 45 ∘ than for 30 ∘ , either with regular or irregular waves (Rosa-Santos et al., 2014). In addition, opposite trends were observed in terms of the evolution of the mean power or the relative capture width with the wave period, for the two tested inclinations. For fixed wave heights, the absorbed power reduces with the wave period for the 45 ∘ angle and increases for the 30 ∘ angle. So, it was concluded that the 30 ∘ angle could become more advantageous for some sea wave conditions (not tested). It was suggested that this behavior could be related with the relationship between CECO inclination angle and the wave steepness. In fact, as the wave height was maintained in the tests, the increase of the wave period corresponds to an increase of the wave length and, therefore, to a wave steepness reduction. The PTO system used in the second study allowed varying the damping introduced in the WEC model, in order to simulate different conditions of wave energy extraction. In general, the amplitude of CECO’s motions increased with the reduction of damping. However, it is important to mention that larger motions do not mean, necessarily, larger power outputs since, in this context, the damping introduced in the system is related to the amount of wave energy converted. The results suggested that the damping in the system influences significantly CECO’s response and that this parameter should be studied in more detail to define the most suitable PTO damping for design wave conditions. CECO model was tested with regular and irregular waves, either short or long crested. The results for regular waves were slightly better than with irregular waves. Possibly due to the small dimensions of this WEC, the use of short crested waves, with a mean direction perpendicular to the alignment of its lateral floating modules, resulted only in a small reduction of the relative capture widths obtained with long crested waves (Marinheiro et al., 2015). However, to accommodate changes in the incoming wave direction, the central element should rotate to modify the orientation of CECO. This adjustment may be done using a controlled mechanical system or a central element with a streamlined profile designed to orientate automatically CECO with the incoming sea waves (Rosa-Santos et al., 2015a). Summing up, CECO performance depends signif icantly on the wave characteristics (wave period and wave height), the PTO damping and the inclination of the device. In the continuity of the experimental proof of concept studies, numerical simulations were used to better understand the CECO response under wave action and to optimize its performance, as detailed next. The preceding studies have already point out several aspects that could contribute to a better performance of CECO: (i) an automatic gear box to reduce the fluc tuations of the PTO rotating velocity, (ii) a system to avoid the inversion of the PTO rotation direction, (iii) a mechanical smoothing system to reduce peak outputs and smooth the power output. In addition, adaptive systems, controlled by incident or predicted wave data, will be needed to maximize the power production in real sea conditions.

4 NUMERICAL MODELLING

4.1 Numerical model description

As mentioned above, Ansys ® Aqwa™ was used to carry out the numerical modeling of CECO with the aim of better understanding its behavior and per formance. This engineering tool suite is commonly applied in offshore and ship engineering, and has been satisfactory applied to simulate the hydrodynamics of different WECs (Bosma et al., 2014, Pastor and Liu, 2014). In a first step, the behavior of the device was repro duced in the frequency domain with a hybrid method based on two approaches. The large-volume structures (i.e. the floating modules and the fixed central cylin der) were modeled as panels through the Boundary Element Method (BEM), which calculates the poten tial flow diffraction and radiation. As for the slender structural elements (i.e. the supporting frame and the longitudinal rods), Morison's equation was used. Both panels and Morison elements were integrated to solve the general equation of motion:

where m = total mass of the moving structure; x = translation of the CECO floating part; F 1 = excitation force, which includes the Froude-Krylov forces and the diffraction forces; F 2 = radiation force; and F 3 = hydrostatic restoring force. After obtaining the excitation forces, added mass, damping matrices and wave field pressure in the frequency domain, the model of CECO was sim ulated in the time domain. Given that the wetted surfaces of the structure change significantly during its performance, the non-linear Froude-Krylov and the hydrostatic forces beneath the incident waves were recalculated and applied to the different parts for each time step. Figure 3. Lateral view of the mesh used for the numerical modeling of CECO. A 1:20 scale model with a 45 ∘ inclination, as tested in the second proof of concept study (described in section 2 and 3), was numerically simulated. A mesh with a spatial resolution of 0.04 ± 0.02 m was built, which, in total, accounted for a total of 2349 panels and 304 Morison elements (Figure 3). The time step of the simulations was set to 0.1 s (model scale). 4.2 Calibration and validation of the model In order to accurately reproduce the behavior of the device, the reaction forces induced by the energy conversion machinery into the oscillating system were considered in the numerical model. These forces include the mechanical losses in the contact movingfixed parts and damping induced by the PTO system itself. To achieve this, a fourth term was added to the dynamic equation (Equation 1), which now reads: where and where f = constant frictional force (f = 0 N, for dx/dt < 0.1 m.s −1); and c = damping coefficient. The parameters f and c were calibrated for two configurations: with and without PTO system. Following an iterative procedure, their values were varied and used in tests performed for different regular wave conditions. For each pair of values and test, time series of CECO motions obtained in the physical and numerical tests were compared and the normalized root mean square error (NRMSE) relative to the numerical model was then calculated. Finally, an error was attributed to each combination of f and cby averaging the NRMSE estimated for the different tests performed.

Table 1. Test conditions considered for the calibration (in prototype).

Test	PTO	H [m]	T [s]	P w [kW]
1	no	1	8	138
2	no	2	8	552
3	no	2	10	664
4	no	2	12	735
5	no	3	10	1495
6	no	3	12	1665
7	yes	1	8	138

8	yes	2	8	552
9	yes	2	10	664
10	yes	2	12	735
11	yes	3	10	1495
12	yes	3	12	1655
13	yes	3	14	1759
14	yes	4	14	3127

Figure 4. Calibration results for the configuration with no PTO.The plus signal indicates the values tested.The red circle indicates the pair of values minimizing the error (values in prototype).

The wave conditions tested correspond to wave heights (H) ranging between 1 and 4 m, and wave periods (T) ranging between 8 and 12 s. The detailed test conditions are summarized in Table 1. P_w represents the average incident wave power, as explained latter.

The configuration without PTO was the first one to be calibrated. Given that the friction and damping forces introduced in the system (Equation 3) in this configuration can be exclusively related to the mechanical losses in the contact moving-fixed parts, the corresponding values of the constant frictional force and the damping coefficient are referred to as f_{loss} and c_{loss}, respectively. As can be observed from Figure 4, the pair of values that minimizes the

error is: f loss = 24.0 kN and c loss = 53.3 kN.s.m −1 , with

NRMSE = 6.6%. Having as reference previous works (e.g., Sheng

et al., 2015, Ricci et al., 2008) and the character

istics of the system used in the proof of concept

tests, the PTO was simulated as a pure damper in Figure 5. Calibration results for the configuration with PTO. The diamonds indicate the values tested (in prototype). the numerical model. Therefore, in the second configuration analyzed, only the damping coefficient was calibrated, and the value of the constant frictional force was maintained equal to the one determined for the configuration without PTO (f loss). Figure 5 summarizes the calibration results obtained, showing a minimum in the error at c = 61.5 kN.s.m −1 with an average NRMSE = 14%. A damping coefficient repre-sentative of the PTO system was obtained as the difference in the values obtained in the two calibrations, i.e. c PTO = 61.5 − 53.3 = 8.2 kN.s.m −1 . 4.3 Captured wave power Once the numerical model was calibrated, the two configurations proposed (i.e. with and without PTO system) were simulated under a set of new regular wave conditions. The 54 tests simulated for each configuration corresponded to the combinations between H = {0.5, 1.0, 2.0, 3.0, 4.0, 5.0} m and T = {6, 7, 8, 9, 10, 11, 12, 13, 14} s (in prototype). On the basis of the results obtained in the simulations, the capability of theWEC to capture wave energy was examined. With this aim, several parameters were defined and calculated for each condition simulated. The average wave power incident on the device during a given test was defined as: where ρ= water density; g = acceleration of gravity; l = total width of the device (in the normal direction to the wave front); and C g = group celerity of the incoming waves. The latter was computed as: where L = wavelength; k = wave number (2π/L); and d = water depth. The average wave power captured by CECO was

defined as (Falcão, 2010):

and the efficiency of the device in capturing this

incident wave power as:

Furthermore, the wave power captured by the device was decomposed in two parts: the average power dis

sipated due to the mechanical losses (P_{loss}) and the

average power delivered to the PTO system (P_{PTO}),

defined as:

and

For the configuration with no PTO system, the val

ues of P_{cap} ranged between 4.6 and 1031.3 kW (in

prototype). These minimum and maximum values

corresponded to test conditions with H = 0.5 m and

T = 14 s, and with H = 5.0 m and T = 11 s, respec

tively. As for the configuration with PTO system,

similar results were found. The minimum and max

imum values of P_{cap} were of 4.8 kW (H = 0.5 m,

T = 14 s) and 1070.8 kW (H = 5.0 m, T = 10 s). As can be noticed, the wave conditions for which

CECO presents the lowest and the highest absorbed

power do not correspond to those with the lowest and

the highest energetic content. This result reveals an

influence of the wave conditions in the ability of this

WEC capturing the wave power. In fact, as mentioned

before, when analyzing the efficiency of the device,

its dependence on the wave height and wave period

becomes evident. For both configurations tested, the efficiency of

CECO presents the same general pattern. The curves

plotted in Figures 6 and 7, which present the efficiency of the device as a function of the wave period and wave

height, present apparent peaks that vary their posi

tions depending on the wave parameters. For H = 0.5,

1, 2 and 3 m, η is maximum when T = 9 s, while

for higher wave heights the peak is located around

T = 10 s (with PTO and H = 5 m) or T = 11 s (without

PTO and H = 5 m). The numerical results also show that the efficiency

of the tested CECO geometry strongly depends on the

wave height. For instance, the maximum efficiency for

H = 0.5 m is approximately 50%, while for H = 5.0 m

it falls to 25%. From the results, it can be stated that including a

PTO system had a relative low influence on the effi

ciency of CECO in capturing wave energy. This may Figure 6. Percentage of the incident wave power captured by CECO with no PTO system (values in prototype). Figure 7. Percentage of the incident wave power captured by CECO with PTO system (values in prototype). be motivated by the relative low value of the damping introduced in the system by the PTO (c PTO) in relation to the damping associated to other mechanical losses (c loss). Finally, the wave power transmitted to the PTO system was analyzed for the set of wave conditions tested. The values of P PTO obtained range between 0.57 kW (H = 0.5 m, T = 14 s) and 131.4 kW (H = 5.0 m, T = 10 s), representing between 9.0 and 12.4% of the energy captured (Figure 8). 5 CONCLUSIONS The numerical modeling of CECO was presented for two different configurations: with and without PTO system incorporated. The Boundary Element Method

Figure 8. Average power captured by CECO and average

power delivered to the PTO system (in prototype).

(which was applied in this work by means of Ansys©

Aqwa™) resulted very satisfactory for carrying out the simulations and allowed reproducing the behavior of the PTO system. Physical modeling results gath ered in previous proof of concept studies were used to calibrate and validate the model.

The forces introduced in the system by the energy conversion machinery, i.e. the mechanical losses and the power delivered to the PTO, were simulated by including a new linear term in CECO dynamic equa tions. Since low errors were obtained when comparing the numerical results with the experimental ones, the approach followed can be considered valid. In addi tion, it allows differentiating the power delivered to the PTO system from the power associated to the mechanical losses.

The efficiency of CECO in capturing the incident wave energy was analyzed and the results obtained are promising: values higher than 50% were found for some test conditions. Nonetheless, to maximize the electric power production of the WEC, the power delivered to the PTO system should be maximized. In order to achieve this, other damping coefficients need to be simulated. Additional topics to be considered in the following development stages include the numer ical modeling of CECO response for irregular waves, the optimization of the lateral floating modules size

and geometry, among others. Summing up, this work has shed light on the perfor

mance of CECO, specially regarding to its efficiency

in capturing wave energy and delivering it to the PTO

system. Moreover, the calibration and the numerical

results presented are a solid basis for future numerical

optimization of this WEC.

Ansys, 2013.Aqwa Theory Manual. ANSYS, Inc., USA, Release 15.0, November 2013. Bosma, B., Sheng, W. and Thiebaut, F., 2014. Performance Assessment of a Floating Power System for the Galway Bay Wave Energy Test Site. Proc. International Conference on Ocean Energy (ICOE), Halifax, USA 6 Nov. 2014. Elwood, D., Yim, S.C., Prudell, J., Stillinger, C., Jouanne, A. von, Brekken, T., Brown, A. and Paasch R., 2010. Design, construction, and ocean testing of a taut-moored dualbody wave energy converter with a linear generator power take-off, Renewable Energy, 35(2): 348–354. Falcão, A.F. de O., 2010. Wave energy utilization: A review of the technologies. Renewable and Sustainable Energy Reviews, 14(3): 899–918. Heller V., 2012. Development of Wave Devices from Initial Conception to Commercial Demonstration, In Comprehensive Renewable Energy, 8, 79–110. Kofoed, J.P., Frigaard, P., Friis-Madsen, E. and Sørensen, H., 2006. Prototype testing of the wave energy converter wave dragon, Renewable Energy, 31(2), 181–189. López, M., Veigas, M. and Iglesias, G., 2015. On the wave energy resource of Peru. Energy Conversion and Management, 90, 34–40. Marinheiro, J., 2013. Optimization study of an innovative wave energy converter, M.Sc. thesis, Faculty of Engineering of the University of Porto, Portugal [in Portuguese]. Marinheiro, J., Rosa-Santos, P., Taveira-Pinto, F. and Ribeiro J., 2015. Feasibility study of the CECO wave energy converter. Maritime Technology and Engineering (ed: C. Guedes Soares & T.A. Santos), CRC Press, 1259–1267. Pastor, J. and Liu, Y., 2014.Power absorption modeling and optimization of a point absorbing wave energy converter using numerical method. Journal of Energy Resources Technology 136(??): 021207. Ricci, P., Saulnier, J.B., Falcão, A. O. and Pontes, M. T., 2008. Time-domain models and wave energy converters performance assessment. Proc. ASME 2008 27th International Conference on Offshore Mechanics and Arctic Engineering, Estoril, Portugal 5–20 June 2008. Richardson, D.S. and Aggidis, G.A., 2013. The Economics of Multi-Axis Point Absorber Wave Energy Converters. Proc. 32nd Inter. Conf. on Ocean, Offshore and Arctic Engineering – OMAE: Ocean Renewable Energy, ASME. Rosa-Santos, P., Taveira-Pinto, F., Ribeiro, J., Teixeira, L. and Marinheiro, J., 2015. Harnessing the kinetic and potential wave energy: design and development of a new wave energy converter, Renewable Energies Offshore: 367–374. Rosa-Santos, P., Taveira-Pinto, F., Marinheiro, J. and Ribeiro J., 2014b. Performance evaluation of an optimized wave energy converter. 5th Int. Conf. on the Application of Physical Modelling to Port & Coastal Protection, 2, 326–335. Rosa-Santos, P., Taveira-Pinto, F., Teixeira, L. and Ribeiro, J., 2015. CECO wave energy converter: Experimental proof of concept. J. Renewable Sustainable Energy 7: 061704. Sheng, W., Alcorn, R. and Lewis, A., 2015. Optimising power take-offs for maximizing wave energy conversions. Proc. 30th InternationalWorkshop onWaterWaves and Floating Bodies, Bristol, UK 12–15th April 2015. Taveira-Pinto, F., Iglesias, G., Rosa-Santos, P. and Deng, Z. D., 2015. Preface to Special Topic: Marine Renewable Energy. Journal of Renewable and Sustainable Energy, 7(6), 061601. Teixeira, L., 2012. Experimental study of a new wave energy converter. M.Sc. thesis, Faculty of Engineering, University of Porto, Portugal, 156p [in Portuguese]. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Pumped hydroelectric energy storage: A comparison of turbomachinery configurations

A. Morabito, J. Steimes & P. Hendrick

Aero-Thermo-Mechanics Departement, Université Libre de Bruxelles, Brussels, Belgium

ABSTRACT: The technical and economic feasibility of Pumped Hydroelectric Energy Storage (PHES) is increasingly under discussion. Its important capital cost may be mitigated by its relevance in balancing the electrical grid. Indeed renewable energy intermittency forces electric power industry to find new solutions for the

regulation of the electric grid. PHES systems are built around hydraulic and electrical machines.The site head and the flow rate primarily define the type of hydraulic machine used. In this paper three configurations are discussed:

traditional applications that include separate hydraulic machines (turbines and pumps), Reversible PumpTurbines (RPTs) and Pumps as Turbines (PATs) that can operate in both modes of operation. A detailed estimation of the PATs performance is computed to predict and interpret their behaviour during reversed operation: therefore different prediction models are listed and analyzed. A case study compares different system efficiencies and capacity for different configurations. This PHES consist in a unused mine with a depth of 350 meters and it is equipped with one or more Francis turbine or with PAT unit. The available head heavily varies with interittency due to the peculiar mine configuration. Consequently the hydraulic machines adopted could often work in off design conditions. Results show that regarding micro and small power plants the solution using PATs is an available and suitable option.

1 INTRODUCTION

In the past decade, the concerns about world envi ronmental issues produced by fossil fuel exploitation have increased. The use of renewable energy sources aims to reduce the dependence on thermal power plants, that nowadays satisfy most of the global energy demand. Among all the technologies usable for this goal, hydro-power is by far the most used in electric energy production. Although hydro-power brings up several problems linked to its social and environmental implications, such as disturbing wildlife and biodiversity or altering landscapes, it has several important positive effects. It maintains a stable price of energy that is very depen dent on political and geographical questions (as the crude oil or the gas market); it is a long-term invest ment and guarantees low emissions for all the working duration with a relatively high efficiency (Barnes 2011). In such context, the Pumped Hydroelectric Energy Storage (PHES) finds its role as the most mature technology regarding energy storage (Deane 2010). PHES systems obviously have many similar ities to conventional hydro-power plants but differ by the fact that the flow is bidirectional. A PHES unit exploits the potential energy stored in the upper reservoir as a conventional hydro-power plant, but it converts electric energy from the grid to refill this reservoir, by pumping back water from the lower reser voir when it is economically profitable. The system working conditions are then designed according to the electricity trading and regulations services needed by the local energy market; for instance, PHES systems dedicated to peak load would be designed for generating, while pumping during light load. To provide a better overview of the possible solutions in a PHES systems, the authors describe the main features of the hydraulic turbines generally installed and their comparison in terms of efficiency and operating range. This paper also gives an overview on the technology of the centrifugal Pumps As Turbines (PATs), proposing a review of the model dedicate to forecast pumps behaviour in reverse mode operation. A non-conventional test case analysis where an existing unused mine of 350 meters deep would be used as a lower reservoir in PHES. The available head heavily varies with interittency due to the peculiar mine configuration. Consequently the hydraulic machines adopted could often work in off-design. 2 TURBO-MACHINES IN HYDRO-POWER PLANTS The first important criterion in order to define a turbomachine for a selected hydroelectric plant is the available head H [m] and the plant power size P [kW] or energy generation [kWh] target (Gulliver & Arndt 1991).

Figure 1. Application chart for hydraulic turbines (Gulliver & Arndt 1991). In general for micro-hydro as shown in Fig.1, the

Pelton turbines cover the high head domain down to

50 m. The Francis turbine covers the largest range of

head under the Pelton turbine domain with some over

lapping and down to 10 m head. The lowest domain

of head for micro-hydro (below 10 m) is covered by

Kaplan turbines with fixed or movable blades. For low

heads and up to 50 m, the cross-flow impulse turbine

is also used. In large power plants the most used turbines are

reaction turbines and are used in low (<40 m) and

medium (40–500 m) head applications (Fig.1). Reac

tion turbines are moved by water, which changes

pressure as it moves through the turbine and gives up

its energy. In reaction turbines loss is generated in both

fixed and moving blades. Impulse turbines are often used in very high head

applications (>300 m). In impulse turbines the jet

pushes on the turbine's blades which changes the direc

tion of the flow. The fluid flowing over the rotor blades is constant and all the work output is due to the change

in fluid kinetic energy.

2.1 Turbo-machines options in PHES

PHES uses different hydraulic machines. Turbines

and pumps, respectively, produce energy (generation)

and consume energy restoring the capacity of the

upper reservoir (pumping). The basic arrangement is

a ternary pumped storage system that consists of a set

of pumps and turbines. It is composed by thus two

hydraulic machines and one electrical machine. Reversible pump-turbines can also be used: the sys

tem consists of a reversible hydraulic machine, a fixed

coupling and a motor-generator for storing and gener

ating energy. In this system there is only one hydraulic

machine, which works as either a pump or a tur

bine (with flow direction reversed), and one electrical

machine, which works either as a motor or a generator.

The reversible set is much simpler than a ternary set. Nevertheless, it has to stand fluid dynamic issues due to its double function and this might raise its costs. Another challenge for the reversible machine is to provide a good efficiency η in partial load. The most used standard pump-turbine is the centrifugal and diagonal types and it must be furnished with special electrical equipment that allows variable speed in order to cover the required operating range in both modes of operation. This is realized by means of a doubly-fed asynchronous generator or full frequency converter connected to a synchronous generator. An alternative solution for PHES is to install diagonal pump-turbines of the Deriaz type, mostly installed in the 1960s. These machines are able to cover a wide operational range, thanks to their adjustable runner blades, without requiring special electrical equipment.

Deriaz machines can offer an alternative solution to the Francis machine in smaller PHES plants with large variation of the required load (Morabito 2014) with the drawback of more mechanical issues. Another possible configuration for a hydroelectric plant is to use a Pump (diagonal or centrifugal) As Turbine (PAT). The advantages are that pumps are robust, mass-manufactured, more readily available and less expensive than manufactured micro/small-hydro turbines (Ramos, Borga, & SimÃ£o 2009). Micro and small hydraulic plants are mentioned because PAT is able to work in turbine mode to the detriment of high efficiency: the performances are far from the best efficiency point (BEP) due to the fixed internal geometry and absence of flow regulation. For this reason PATs are not recommended for big and medium size plants where a single percentage point of efficiency has a strong impact on the economic feasiblity of the installation (Orchard & Klos 2009) (Derakhshan & Nourbakhsh 2008b). A single installation is not efficient over a wide range of flows; having multiple PATs of different sizes, each optimized for a different flow and suited for specific flow regimes, can overcome this limitation. PATs are studied in more details in this paper.

3 PUMP AS TURBINE

The use of pumps as turbines has been a research topic for over 80 years when engineers accidentally found that pumps were able to operate very efficiently in turbine mode when they were trying to evaluate the complete characteristics of pumps (Kittredge & Thoma 1931, Knapp 1937). In the 1950s and 1960s, the concept of PHES plants evolved mainly in devel oped countries to manage peak power requirements and support nuclear power plants design. In the later years chemical industries became another area for the application of PATs. In certain chemical processes it was necessary to dissipate the energy of high pressure fluids through small pipe lengths. Instead of simply throttling, PATs were installed to recover some energy. Small Hydroelectric (SH) power stations became attractive to generate electrical energy with the increasing interest in a smart use of the energy. How ever the cost per kWh of these power plants was higher than for hydroelectric power plants with large capac ity. Therefore using PATs emerged as an attractive and important alternative. Due to the large market for pumps of all possible capacities it is easily available, cheap and reliable (Orchard & Klos 2009). There fore the maintenance has many advantages compared to custom-made turbines. Pumps are relatively sim ple machines, easy to maintain and readily available in most developing countries. From an economical point of view, it is often stated that pumps working as turbines are profitable in the range of 1 to 500 kW (allowing capital payback periods of two years or less which is considerably less than that of a conventional turbine (Paish 2002)). Pump manufacturers do not normally provide the characteristic curves of their pumps in reverse opera tion. Therefore, establishing a correlation enabling the switch from the pump characteristics to turbine char acteristics is the main challenge in using a PAT. The hydraulic behaviour of a pump rotating as a turbine changes. In general a pump will operate in turbine mode with a higher head and discharge for the same rotational speed. Many researchers have presented some theoretical and empirical relations for predict ing the PAT characteristics in the best efficiency point (BEP). Most recent attempts to predict the perfor mance of PATs, have been made using Computational Fluid Dynamics (CFD) simulations. However, without validating the CFD results by experimental data, they are not fully reliable. Besides, all these simulations included only hydraulic losses. Many prediction techniques have been published.A

few of the early contributors to these techniques were Kittredge (Kittredge 1945) and Stepanoff (Stepanoff 1957). In the later years many more techniques were developed by many researchers namely Nautiyal (Nautiyal 2011) Sharma (Sharma 1985), Schmiedl (Schmiedl 1988), Grover (Grover 1980), and more recently by Williams (Williams 1995), Alatorre-Frenk (Alatorre-Frenk & Thomas 1990) and Cohrs (Cohrs 1977). There are many uncertainties associated with the various prediction methods nevertheless they have served as a starting point in recent disseminations investigated for instance by Sigh and Nestmann (Singh & Nestmann 2010) (Singh & Nestmann 2011) and by Derakhshan and Nourbakhsh (Derakhshan & Nourbakhsh 2008b). 3.1 Performance prediction of pumps in turbine mode The PAT behaviour is very complex and it is difficult to find a unique corelation for all pumps in reverse mode. Several problems have to be solved, linked to hydraulic losses in the volute and the impeller, mechanical losses related to packing and bearing cases, disc friction losses in gaps between rotor and stator and volumetric losses related to leakage from clearances between rotor and stator (Derakhshan & Nourbakhsh 2008b). One of the serious problems of PAT technology is to individuate the right best-input parameters for predicting the performance of the rated pump. Authors have published on the use of PATs in which the turbine performance is predicted using the values of head and flow at the pump's best efficiency point, obtaining scaled values of head and flow rate values in turbine mode. The different formulas, which have been proposed by the various authors, are considered in the following sections. When a pump operates in the turbine zone, the motor will operate as a generator. During pump or turbine operations the discharge, Q, is a function of the rotating speed, N, and the pumping head, H, whereas the alteration of the speed will depend upon torque, T. In order to characterize the machine, power P and efficiency η have to be identified. The relationships between some of these parameters in pump and turbine mode are presented in dimensionless form through the ratio with operational condition: with t for turbine and p pump. Different methods can be used to estimate the best efficiency point in turbine operation from the manufacturer's pump performance data. One of the earliest available paper that presents such a method was published in 1962 by Childs (Childs 1962). He stated that the turbine best efficiency and pump best efficiency for the same machine are approximately equal. He further assumed that the turbine output power for best efficiency operation was equal to the pump input power. Hence: and

It is also assumed that Q t Q p = H t H p and hence Stepanoff (Stepanoff 1957) also proposed a method

that depends on the pump efficiency. It relates the

ratios of turbine and pump head and flow to the

hydraulic efficiency of the pump, by the following

equations: Since the hydraulic efficiency is not normally

known, the following simplification is incorporated: Sharma (Sharma 1985) has developed a prediction

method (preferred by Williams (Williams 1994)), that

also uses ratios dependent on the pump efficiency. He uses the initial assumptions, as in the method presented

by Child (Childs 1962), that $P_p = P_t$ and $\eta_p = \eta_t$, which

results in equation (3) and relates the two specific

speeds for pump and turbine operation by

where specific speed is defined by $n_{st} = N_t \sqrt{Q_t} H_t^{0.75}$ The

following equations can thus be derived:

The method presented by Alatorre-Frenk (Alatorre

Frenk & Thomas 1990) is based on fitting equations to

a limited number of PAT data. The equations are again

based on the pump efficiency, and are expressed in the

form Several other authors have proposed equations that

relate the head and flow ratios for pump and turbine

operation to the pump or turbine specific speed.

The method by Grover (Grover 1980) is restricted

to the specific speed range $10 < n_{st} < 50$: Hergt's method is presented in graphical form in

(Lewinsky-Kesslitz 1987), which shows a range of

head and flow ratios, each given as a function of turbine

specific speed. Hergt's equations are As showed by different experimental researches, the results obtained by these relations deviate almost ±20% from experimental data (Williams 1994). As described by Nestmann and Sigh (Singh & Nestmann 2010) the water inlet angle to the impeller in reverse mode is equal with volute angle. In fact, the volute operates as a guide channel. Assuming there is no swirl effect at the pump discharge, the water outlet angle from the impeller in turbine operation is equal with the impeller inlet angle. So we can consider the same Euler heads for turbine and pump modes: $H_{pEuler} = H_{tEuler}$. Due to the slip of finite blade number, pump and turbine theoretical head can be written as: Where μ is the slip factor for pump operation $\mu < 1$, and ν is the slip factor for turbine operation. The slip factor for reverse mode is approximately equal to 1.0 (Chapallaz, Eichenberger, & Fischer 1992). Considering its hydraulic efficiency, it can be written: η a p = η ph η th , b = 1 μν with a and b bigger than 1 Since it is possible to affirm that v 2 ∝ H , and including the direct and reverse flow with their leakages Q p,l ,Q t,l The best fit from the experimental data brings to rewrite the chosen equations as: with a = c = 1.1 and b = 1.2 3.2 Proposed predicted model Derakhshan and Yang (Derakhshan & Nourbakhsh 2008a) (Yang, Derakhshan, & Kong 2012) have modeled a more complex method using experimental data to predict the best efficiency point of a pump working as a turbine based on pump hydraulic characteristics. In this study, a mini hydro-power test-rig was installed, and four centrifugal pumps (n s < 60 (m,m 3 /s)) were tested as turbines. Experiments showed that a centrifugal pump can appropriately operate as a turbine in various rotational speeds, heads and flow rates. A PAT works in higher head and flow rate than those of the pump mode at the same rotational speed. Efficiency is almost the same in both pump and turbine modes. Combining the experimental data provided by other autors (Derakhshan & Nourbakhsh 2008a, Joshi, Gordon, & Holloway 2005, Singh 2005, Williams 1992)

Figure 2. Sketch of the mine structure: it is composed mainly by two vertical pits that are crossed by two large tunnels. Test

cases: A) one Francis turbine located at 350 m depth; B) Two Francis turbines located at 350 m ad 200 m depth; C) One PAT

at 200 m.

it is apparent by tests that at the same specific speed

different values for h and q may be obtained (Yang,

Derakhshan, & Kong 2012). This leads to the conclu

sion that for smaller values of n sp , the model based

solely on the classification of the specific speed can

not make a perfect prediction (Fig.3). It is possible to obtain a better correlation of pre

diction than other various methods if one collects subgroups defined by impeller diameter. Searching for

a trend line of this model through all the tests reported

with the same diameter (i.e. diameter about 0.250 [m]),

the following equations are found:

PAT efficiency value floats between the value in pump

mode proposed by the most optimistic models men

tioned before and more a mitigated approximation as

the following equation by Gulich (Gulich 2007):

3.3 Variable speed

Units with variable speed are more used, especially

for Reversible Pumps Turbines (RPTs). RPTs are

hydraulic machines designed to work either as a pump

or as a turbine depending on the direction of rotation.

Furthermore, a well-designed, compact power house

save equipment and civil costs. The use of variable speed in PHES represents a

valuable option to deal with off-design working con

dition. These systems couple the motor/generator to

a frequency changer enabling a wider variable speed

pumping or generating (±12%) (Thoni & Schlunegger

2009). Compared to a conventional configuration at a Figure 3. Experimental value of h and q. constant speed, the variable speed offers several advantages. It ensures a rapid response to a daily unsteady demand of electrical power. This technology ensures to combine in a single machine a multiplicity of different design configurations.Adjusting the rotation speed allows to operate as close as possible to an optimal condition. However, the characteristic curve or set of characteristic curves for any turbo-machine can not be determined only theoretically (Round 2004). Better results are obtained by using empirical data available from performance tests at various speeds. Moreover, one must always consider that the calculations include some errors. The reference of -5% to +10% defines the range of rotational speeds where the affinity laws are applicable for pumps (Schoenung 2003, Nourbakhsh, Jaumotte, & Hirsch 2008). The affinity laws are very useful in small speed changes, but they have become predictive tools for VFDs (Variable Frequency Devices) operation where changes may be more dramatic.

Figure 4. Volume of stored water in the mine of the

presented test case.

Figure 5. Efficiency curve for the Francis turbine selected

and estimation of the performance of the PAT with variable

speed. Nevertheless, there are operational speed limits due

to a physical boundary of operation (e.g. cavitation,

vibrations).

4 CASE STUDY

The case study presented is based on an unconven

tional solution for PHES: the lower reservoir is an old

mine traversed by a multitude of pits and tunnels. As

shown in Fig.2, it mainly consists of two main horizon

tal tunnels (about 14700 m 3 each) connected by two

wells (about 12500 m 3 each). The mine extends over a

depth of 500 meters (Archambeau & Erpicum 2015)

but in this case the maximum available head used is

limited to 350 meters. Due to its relevant depth, it is

possible to outrange the nominal design conditions in

power generation mode, generally limited in a small

range of available head. The mine structure is sim plified by secondary minor tunnels and not relevant

cavities. The upper reservoir is located at ground level. In fact, when the gallery on the bottom is filled

(reaching a volume of about 4900 m^3), the wells will

be refilled, which, due their vertical extension, quickly

reduce the available head. At the depth of 200 meters

there is another tunnel. The water volume increases

considerably when the principal tunnels are reached

(Fig.4) at 350 meters and 200 meters depth (Fig.2). Figure 6. Efficiency trend of the turbines while generating and filling the mine. The red square marks the switch of working turbine in the configuration B. It is a synthetic case that can be used for preliminary pumping or turbining phases study. Mines have unlikely a good geometrical description, especially for the most ancient: in fact they much more secondary tunnels and wells for ventilation or additional services. Moreover, the soil has its porosity which, acting as a large buffer and prohibiting the total filling of the mine, affects the process of pumping and generating. Therefore three turbomachinery options are studied for this site can be summarized as following: A) a single power station located on the bottom of the lowest tunnel equipped with a Francis turbine; B) option A with a second Francis unit at 200 m depth ready to operate once the first is in extreme offdesign condition; C) using only a centrifugal pump in turbine mode with variable speed. 4.1 Scenario A In the case with a system using only a single powerhouse at the bottom of the reservoir, a Francis turbine is selected for a nominal head of 350 m. The flow rate used is fixed to 7 m^3/s in order to produce around 20 MW according to the project goals. With this flow rate value the working time is about 36 minutes before the efficiency drops under the 60% due to varying head. This power plant set (configuration A Fig.2) has to deal with the turbine variable efficiency as the head changes with the filling of the mine. The available head decreases while the reservoir is getting full, faster while the vertical pits are crossed (Fig.4). The time ranges where the efficiency looks almost constant occur when the large tunnels are being filled and, consequently, the head remains stable (Fig.6). 4.2 Scenario B The second scenario aims to take advantage of the specific configuration of the mine (Fig.2). Two units have been designed: one with a

Francis turbine at nominal head of 350 meters (FrancisD350) and a second unit designed for 200 meters (FrancisD200), both ready to

better exploit the stationary head condition provided

by the large buffering of the tunnels. A switch point

is found on their variable global efficiency. It defines

the moment where FrancisD200 efficiency is higher

than FrancisD300 efficiency (Fig.6).At this instant the

FrancisD300 shuts down while the second one starts:

this allows the system to keep high efficiency where

a system of the one machine would be in extreme off

design conditions. Hence the global efficiency of the

configuration stands mostly above 90% (Fig.6).

4.3 Scenario C

Despite the fact that the predicted power capacity for

this case study is out of the recommended range for

PATs (Derakhshan & Nourbakhsh 2008a, Paish 2002),

this solution could represent an important alternative.

A single stage commercial pump, matching the condi

tions requested in turbine mode for an available head of

350 meters, has not been found by the authors. A mul

tistage radial flow may not fit in the model discussed

before: the behaviour of the first stage of the PAT will

trickle down to all the following stages, amplifying the

effects and increasing the model uncertainty. Hence a

PAT for an available 200 meter head is chosen. The centrifugal pump selected has an impeller of 980 mm

diameter and a discharge section of 700 mm. Based

on the selected prediction model discussed before

(Par.3.2) and according to the characteristic curves

provided by the pump manufacturer, its behaviour in

turbine mode is predicted. In order to maintain suf

ficiently high efficiency for large range of load, its

rotation speed is changing in the range of 1000-850

rpm thanks to a VFD. Unlike the previous scenarii, in this option the flow

rate is depending on the variable available head accord

ing to trend-line of estimated characteristic curves,

given by the equation (18): In contrast to a conventional hydraulic turbine, a

PAT is not equipped with movable guide vanes, hence

the profile of the flow rate with partial load differs from

a Francis turbine. According to equation 18 the flow

rate drops from the highest value of 5,5 m^3/s to 2,1 m^3/s

and this justifies also the reduced power production

compared to the previous scenarii (Fig.7).

5 CONCLUSIONS

In this report important overviews are described

regarding the possible hydraulic machinery configu

rations used in PHES. The most used but also the most

customized and expensive is the reversible machines.

PATs can offer an interesting option and, although they do not ensure a high efficiency, they have a lower capi

tal cost.The possible efficiency detriments of the PATs

might be mitigated and in cases off set by the detailed
Figure 7. Power generated by the different configurations.
research conducted on forecasting PATs behaviour and thanks to VFD. PATs show a value of power generation reduced compared to the other options, due to variable mass flow rate over time. Concerning the efficiency, PATs supported by theVFD are able to work more efficiently than turbines in the relevant off-design range. The performance of hydraulic turbines appears to be higher than PATs as shown in Fig.5 and Fig.6. Regarding to the market, accessibility and maintenance, dealing with a hydraulic centrifugal pump is categorically simpler and economically more slender (Ramos, Borga, & SimÃ£o 2009). The pump market is well developed and several experiences have shown that centrifugal pumps can operate in turbine mode without any important difficulties (Carravetta & Ferracotta 2014, Orchard & Klos 2009, Ariaga 2010). Compared to a ternary pumped storage system, PATs, as RPTs, have the advantages of saving room and the consequent excavation costs.That is an important issue for hydroelectric plants with power-house built deep in the ground as in the case study. The configurations proposed for the case study had to demonstrate the profitability of this nonconventional PHES plant. While the configuration B can properly operate in steady state conditions for a longer period of time than the other options, the configuration C guarantees a relevant energy capacity for the mine exploitation with the most reduced investment among the configurations. Therefore the integration of two options may occur, giving as result the installation of a Francis turbine designed for 350 meters head and a PAT operating at 200 meters depth. It would be worthwhile detecting all the structural advantages of working with generating units located at different available heads. Pipelines and excavation costs have been voiced in this regard. If this analysis finds success, it will be reasonable to design even a third power-house deeper due to the fact that the selected mine reaches deeper. This arrangement might be an example to laud the profitability of the PHES plants and revaluation of spaces unused. PHES system have all the features needed for being a suitable solution for the impelling requests of smart use of energy. There are still questions regarding the

economic profitability of these applications and their social and environmental effects that still happens for hydro-power. This paper gives an overview of the configurations and settings for PHES even in a non-conventional case and enables further issues and challenges for the future.

ACKNOWLEDGEMENT

This research was supported by Region Wallonia.

The data presented, the statements made and the views expressed have been achieved thanks to the SmartWater project.

Alatorre-Frenk, C. & T. Thomas (1990). The pumps as turbines approach to small hydropower. World congress on Renewable energy, Reading.

Archambeau, P. & S. Erpicum (2015). Coordination wp4 wp5. Technical report, Projet Smartwater, Liége.

Ariaga, M. (2010). Pump as turbine a pico-hydro alternative in lao people's democratic republic. Renewable Energy 35, 1109–1115.

Barnes, F. (2011). Large energy storage system handbook. New York: CRC press.

Carravetta, A. & O. Ferracotta (2014). Energy recoverying water system by pats: a comparison among different installations scheme. Procedia Engineering 70, 275–284.

Chapallaz, J., P. Eichenberger, & G. Fischer (1992). Manual on pumps used as turbines. Braunschweig: Vieweg.

Childs, S. (1962). Convert pumps to turbines and recover hp. Hydro Carbon Process Pet Refin 41(10), 173–4.

Cohrs, D. (1977). Investigations of a multistage pump operated as a turbine. Untersuchungen an einer mehrstufigen ruckwartslaufenden kreiselpumpe im turbinenbetrieb.

Deane, J. (2010). Techno-economic review of existing and new pumped hydro energy storage plant. Renewable and

Sustainable Energy Reviews 14(4), 1293–1302.

Derakhshan, S. & A. Nourbakhsh (2008a). Experimental study of characteristic curves of centrifugal pumps working as turbines in different specific speeds. Experimental

Thermal and Fluid Science 32(3), 800–807.

Derakhshan, S. & A. Nourbakhsh (2008b). Theoretical, numerical and experimental investigation of centrifugal pumps in reverse operation. Experimental Thermal and Fluid Science 32(8), 1620–1627.

Grover, K. (1980). Conversion of pumps to turbines.

Gulich, J, F. (2007). Centrifugal pumps. Verlag Berlin Heidelberg: Springer.

Gulliver, J. & R. Arndt (1991). Hydropower engineering handbook. McGraw-Hill.

Joshi, S., A. Gordon, & L. Holloway (2005). Selecting a high specific speed pump for low head hydro-electric power generation. Electrical and Computer Engineering Canadian Conference, 603–606.

Kittredge, C. (1945). Centrifugal pumps used as hydraulic turbines. pp. 59–66. Kittredge, C. & D. Thoma (1931). Centrifugal pumps operated under abnormal conditions. Power HYD-59-11, 881–884. Knapp, R. (1937). Complete characteristics of centrifugal pumps and their use in the prediction of transient behavior. ASME, 683–689. Lewinsky-Kesslitz, H. (1987). Pumpen als turbinen fur kleinkraftwerke. Wasserwirtschaft 77(10), 531–537. Morabito, A. (2014). Design Method and Optimisation of Deriaz Pump-Turbine for hydraulic energy storage. Milan, Italy: Politecnico di Milano and Universitité Libre de Bruxelles. Nautiyal, H. (2011). Experimantal investigation of centrifugal pumpworking as turbine for small hydropower system. Energy Sci Technol 1, 79–86. Nourbakhsh, A., A. Jaumotte, & C. Hirsch (2008). Turbopumps and pump system. Springer. Orchard, B. & S. Klos (2009). Pumps as turbines for water industry. World Pumps 2009(8), 22–23. Paish, O. (2002). Micro-hydropower: Status and prospects. Proceedings of the Institution of Mechanical Engineers, 216, 31–40. Ramos, H. M., A. Borga, & M. SimÃ£o (2009). New design solutions for low-power energy production in water pipe systems. Water Science and Engineering 2(4), 69–84. Round, G. (2004). Theory of Turbomachines, In Incompressible Flow Turbomachines. Ontario, Canada: Burlington. Schmiedl, A.

(1988). Serien-Kreiselpumpen im Turbinenbetrieb. Pumpentagung Karlsruhe. Schoenung, S. (2003). Long vs Short Term Energy Storage Technologies, Analysis. Sandia report. Sharma, K. (1985). Small hydroelectric projects-use of centrifugal pumps as turbines. Bangalore, India: Kirloskar Electric. Co. Singh, P. (2005). Optimization of internal hydraulics and of system design for pumps as turbines with field implementation and evaluation, p.h.d.m.e. thesis. Singh, P. & F. Nestmann (2010). An optimization routine on a prediction and selection model for the turbine operation of centrifugal pumps. Experimental Thermal and Fluid Science 34(2), 152–164. Singh, P. & F. Nestmann (2011). Internal hydraulic analysis of impeller rounding in centrifugal pumps as turbines. Experimental Thermal and Fluid Science 35(1), 121–134. Stepanoff, A. (1957). Pumps as Turbines Users Guide. New York: John Wiley. Thoni, H. & A. Schlunegger (2009). 100 MW fullsize converter in the Grimsel 2 pumped-storage plant. Innertkirchen. Williams, A. (1992). Pumps as turbines used with induction generations of stand-alone micro-hydroelectric power plants, Ph. D. Thesis. Williams, A. (1994). The turbine performance of centrifugal pumps: A comparison of prediction methods. Power and Energy. 208, 59–66. Williams, A. (1995). Pumps as Turbines Users Guide. London: International Technology Publications. Yang, S.-S., S. Derakhshan, & F.-Y. Kong (2012).Theoretical, numerical and experimental prediction of pump as turbine performance. Renewable Energy 48, 507–513. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Cost and revenue breakdown for a pumped hydroelectric energy storage

installation in Belgium

J. Steimes, G. Al Zohbi & P. Hendrick

Aero-Thermo-Mechanics Department, Université Libre de Bruxelles, Brussels, Belgium

B. Haut & S. Doucement

Mathematics, Data Processing and Information Technology & EGEU, Laborelec, Linkebeek, Belgium

ABSTRACT: The technical and economic feasibility of Pumped Hydroelectric Energy Storage (PHES) is

increasingly under discussion in Belgium and worldwide. The

important initial capital cost, a barrier to the use of PHES systems, could be counterbalanced by using pre-existing installations (as mines or quarries). However, it is necessary to have an estimation of the costs and revenues breakdown of these installations in order to optimise the number and the type of components that will be installed. This paper presents the cost and revenue breakdown for a PHES installation to be installed in Belgium, more specifically in the Walloon region. It presents the division of the plant into sub-parts for which the investment costs are modelled (reservoirs, electrical lines, etc.). An optimisation of the revenue sources available to PHES plants in Belgium is then presented. This helps to optimise the utilisation of the plant. Operation and Maintenance costs are finally taken into account to build a financial analysis of the investment. These computations help to estimate the most promising installation for a dedicated site. This is then illustrated as a conclusion on a case study of a PHES site of 4MW and 20MWh installed in a quarry and a mine. Results show that the installation in the mine seems more promising, but specific additional startup costs will have to be taken into account in futures works.

1 INTRODUCTION

1.1 Context

Pumped Hydroelectric Energy Storage (PHES) plants store energy in the form of potential energy of water that is pumped from a lower reservoir to a higher level reservoir. In this type of system, low cost electric power (electricity in off-peak time) is used to run the pumps to raise the water from the lower reservoir to the upper one (Fig. 1). During the periods of high power demand, the stored water is released through hydro turbines to produce electric power. PHES has many advantages. Current pumps/turbines have the capability to work in all possible modes of operation, under full automatic control with auto matic operation of transient states and quick changes between them (1–5 min).They can be easily controlled remotely, have high start/stop frequency and the high est availability and capability to support black starts. Innovations in variable-speed motors have helped these plants to operate at partial capacity and have greatly reduced equipment vibrations. In an integrated system, storage helps to reduce the challenges of inte grating variable renewable resources (Krajacic & Al. 2013). For this goal, PHES is the most mature tech nology and presents the largest capacity (Ding & Song 2012). Figure 1. Pumped Hydro Electric Storage system schematic. Pumped storage plants are characterized by long construction times and high capital expenditure. The price of new installations can vary between 500 and 3600 €/kW (Krajacic & Al. 2013). This dispersion is explained by the fact that PHES is a resource driven facility which requires very specific site conditions to make a project viable, i.e. high head, favourable topography, good geotechnical conditions, access to electricity transmission networks and water availability. This generates very different and specific sites for which the capital costs can greatly vary. In Belgium, the only two PHES facilities are located in Coo and Platte Taille. They have a total capacity of respectively 1100 and 143 MW (produced for 5 hours).

Initially built in order to regulate electricity generation coming from nuclear reactors, they are now

Figure 2. Technical-economical analysis methodology.

increasingly used as a tool to balance generation and load on the grid.

There are not many promising sites providing suffi cient head and space for conventional PHES storage in Belgium. Therefore, unconventional applications that use quarries and mines as storage reservoirs are stud ied and might lead to lower capital costs due do the (partial) availability of the reservoirs. Generally, their capacity and generation will be lower than in conven tional sites. As the financial viability of such sites is closely linked to their initial capital investment, their technical-economical analysis requires specific costs and revenues breakdown that help determine an optimal site configuration and maximize the revenues.

1.2 Content and methodology

The financial viability of a PHES site is closely linked to its capital costs, its operating costs and the revenues it generates. This paper presents this cost and revenue breakdown applied to two new configurations (quarry and mine). Details are given on the methodology and correlations used. Finally, a case-study illustrates the impact of the location and site configuration on these effects and the difference between the use of mines and quarries. The methodology followed in this study is summarized on Fig. 2. Input variables reflect the specificities of the stud ied site, and are usually source of CAPEX variations. For instance, it takes into account the type of reser voir (mine, quarry, artificial), the required excavation and coating for the tanks and the machine rooms. The number and type of hydro turbines and the distance to the electrical network are considered. Finally elec tricity prices and ancillary services can also be finely tuned.

These inputs feed a capital cost study (which is thus specific to one site configuration). The performance of the system are then computed and used to compute optimal revenue combination. This is used to defined operation and maintenance costs and the Net Present Value (NPV) of the plant. This last parameter is used to assess the financial viability of the installation.

2 COST BREAKDOWN

The capital cost covers several different parts of the installation. This study focuses on a detailed study of a variety of PHES potential sites. Therefore, the cost break-down is extensive and takes into account an important number of parameters. They are separated in four categories:

- Electro-Mechanical (EM): linked to the choice of pumps and turbines, the length of the electrical lines and their power, electrical installation parts, etc.
- Civil Works : excavation, using existing tanks (mines, quarries), their permeability to water (coating

requirements), the spillways and pen stock costs, etc. • Operation and Maintenance: Due to the operation of the plant and the use of the components. • Indirect costs : regroups engineering, development and unpredicted costs. A number of these costs correlations is already presented in the literature, others are computed for this study. They are summarized in this section. 2.1 Electro-mechanical The selection of EM components depends on the desired performance and technical characteristics of the site. Two are discussed here: the hydro turbines and the electrical connections. 2.1.1 Turbines PHES site can be operated with one hydro turbine acting as a pump and as a turbine or can be composed of two hydro turbines, one dedicated to each function. When considering a combination of reversible turbines and pump/turbine, three different turbines are generally used: Kaplan, Francis and Pelton turbines. Selection of an appropriate turbine is dependent to a large extent upon the available water head and to a lesser extent on the available flow rate.Their efficiency and net head also vary. The selection is particularly critical in low-head schemes, where large discharges need to be handled to be economically viable. In general, impulse turbines are used for high head sites, and reaction turbines are used for low head sites. Kaplan turbines with adjustable blade pitch are suitable for a wide range of flow or head conditions, since their peak efficiency can be achieved over a wide range of flow conditions. Small turbines (less than 10 MW and for some fairly large bulb-type turbine to 100MW) can have horizontal shafts. Very large Francis and Kaplan machines usually have vertical shafts because it makes best use of the available head, and allows a more economical installation of the generator. Pelton turbines may be installed either vertically or horizontally (Paish 2002). A significant factor in the comparison of different turbine types is their relative efficiencies both at design point and at reduced flows (Typical efficiency curves are shown in (Paish 2002)). The Pelton and Kaplan turbines retain very high efficiencies when running below design flow; in contrast the efficiency of the Crossflow and Francis turbines is lowered if used under their nominal flow rate. Most fixed-pitch propeller turbines

perform poorly except above 80% of full flow rate

(Paish 2002). Pumps can be operated as turbine (PAT) to gener

ate electricity by changing the mode of operation and

reversing the water flow direction. A primary advan tage of using a PAT instead of a classical hydro turbine is the potential cost savings. Due to the large num ber of standard pumps produced, a pump converted to a turbine can be significantly less expensive than a specifically designed hydraulic turbine. In addition to low cost, PAT has several other advantages. These include simplified control due to the absence of blade pitch change, simplified installation, ready adaptation to chemical or hot process fluids, and reduced delivery time. These factors can also contribute to cost savings (Hossain, Ferdous, Salehin, Saleque, & Jamal 2014). The cost of electro-mechanical equipment can rep resent a high percentage of a small hydro-power plant budget (around 30% and 40% of the total sum (Zhang, Smith, & Zhan 2012)).This cost can thus directly influ ence the project feasibility (Ogayra & Vidal 2009). Several authors studied the cost of classical hydro tur bines (Kaldellis, Vlachou, & Korbakis 2005), (Ogayra & Vidal 2009),(Agadis, Luchinskaya, Rothschild, & Howard 2010),(Zhang, Smith, & Zhan 2012). In this paper, the results of Papantonis (Papantonis 2001) are used. They estimate the costs of different com ponents of the hydro plants based on European data. This includes the formulae to estimate the costs of electro-mechanical equipment (turbine, speed control and generator), cost of different types of turbines (Kaplan, Francis and Pelton), cost of generators, speed controls. They are summarized in Fig. 1 and Tab. 1. To be used in this study, an inflation factor between years 2001 and 2015 is used.

with C_{EM} the cost of the electro-mechanical equipment in € 2015 , P the power in kW and H the head in m and I the inflation rate between 2001 and 2015.

2.1.2 Electrical lines

In Belgium, electrical lines installation price depends on the power, the supply voltage level and the length of the connecting line. For low and mid voltage powers, they are provided by the DSO (Ores 2015). For higher voltages, the prices are fixed by the TSO. They are mainly represented by Eqn. 2 and Tab. 2

with L_{elec} the length of required electrical wires in meters and $K_{1,2,3}$ given in Tab. 2.

2.2 Civil works and tanks

The cost of civil engineering includes the cost of the reservoir, pen stock, installation of the machines and reservoir (which includes the cost of the land, of the excavation and the coating). Table 1. Cost parameters for hydro turbines (Papantonis 2001). Type K 1 K 2 K 3 Kaplan 35 446 0.410 -0.2100 Francis 33 676 0.481 -0.2858 Pelton 43 465 0.444 -0.1858 Table 2. Electrical line prices, data compiled from (Ores 2015). P(MW) 0-5 5-8 8-10 10-16 16-20 K 4 38.09 8.32 7.79 6.08 5.71 K 5 44 110 110 110 110 K 6 6548 2026247 207591 30623 307798 2.2.1 Reservoir Fundamentally, a reservoir serves to store water and the size of the reservoir is governed by the volume of water that must be stored. The construction of a reservoir depends on several parameters and on the kind of proposed tank. In this study man-made reservoir are studied in the form of • Open air cavities, as some quarries. • Underground cavities, as some mines. The reservoirs should be as tight as possible to avoid groundwater contamination and any water losses. This will depend on the material of the cavity as some are permeable and others are watertight.This study aims to take advantages of existing cavities (quarries, mines), but one of the two reservoir may have to be constructed from scratch.As a general rule, the actions necessary to use a cavity for PHES can thus be: excavation, coating, land purchase and spillway construction. The cost of these different parts are estimated in Eqns. 3 to 6.These formulas are based on current prices for similar civil works in Belgium. with C land the cost of the land in € 2015 , S tank the ground surface of the tank in m 2 , C exca the cost of excavation in € 2015 , V exca the volume that needs to be excavated in m 3 , C coat the cost of coating in € 2015 , S coat the surface that needs to be coated in m 2 and C sp the cost of the spillways in € 2015 . 2.2.2 Pen stock and piping The pen stock leads the water from the intake to the turbine. It is constructed to create a high head with the shortest possible distance to reduce losses and investment costs. The most commonly used material for pen

Table 3. Installation costs of a penstocks (Andaroodi & Al.

2005). Slope ≤ 10% Slope > 10%

C pi (€ 2015 /m) 200 + 140 D 400 + 140 D

stocks pipe is steel. PVC can also be used and can help

to reduce investments.

The penstock investement cost is computed based

on the material cost and on the installation costs.

Material cost can be estimated by using the weight

of required material (linked to the thickness, diameter

and length of the pipe).The optimal diameter regarding

pressure losses is given by Eqn. 7 (Fablbusch 1987).

with D the pipe diameter in m, Q the maximum flow rate in the pipes in m^3/s and H the available head in m.

The thickness is defined by the water pressure in the pipe (due to the head H), and by permissible stresses in the piping material and joints. It has to withstand water hammer in case of sudden closure of the turbine. This is summarized in Eqn. 8

with t_p the pipe thickness in m and σ the permissible stress in the pen stock material in MPa. From Eqns. 7 and 8, and from the length of the pipes L_p, the weight of required material can be computed as in Eqn. 9

with L_p the pipe length in m and ρ the density of the material in kg/m^3. Installation costs are defined in Tab. 3, they depend on the slope and on the diame ter of the pipes (excavation, etc.) (Ogayar, Vidal, & Hernandez 2009). Finally, the cost of the pen stock is defined as

with C_P the cost of pen stock materials and installation in € 2015 and C_{mat} the cost of material per kg.

2.3 Operation and maintenance

The plant maintenance ensures reliable and uninter rupted power supply. This leads to yearly Operation & Maintenance (O&M) costs that are due to:

- Maintenance of electrical systems: generators, transformers, switch yard equipment, station aux iliaries, excitation system, controls, metering, pro

tection systems etc. • Maintenance of civil structures: barrage, head regulator, de-silting basin, power channel or tunnel, fore bay, surge tank, tail race channel, power house building, switch yard foundations and trenches, etc. • Maintenance of hydro-mechanical components: turbine, barrage gates, head regulator gates, intake gates, spillway gates, trash racks, expansion joints, etc. Usually, O&M costs are taken as a fraction of the capital cost (1.5 to 2% in (Zhang, Smith, & Zhan 2012)). They can be divided in two parts: the fixed maintenance costs and the variable ones that depend on the utilization of the plant. Connolly et al. estimated the total O&M cost as a value of the fixed O&M costs (C OM ,f), depending of the total installed power, and on the variable cost (C OM ,v), depending on the electricity production, as given by Eqn. 11 and 12 (Connolly & al 2011). 2.4 Indirect costs The indirect costs include the engineering costs such as permitting applications, project management and the contingency costs. They represent up to 50% of the direct costs (costs of EM and civil works) (Zhang, Smith, & Zhan 2012). 3 REVENUES BREAKDOWN AND COMPUTATIONS PHES plant operation was traditionally limited to ancillary services and power backup. However, the evolution of the electrical network led to use them for other network services. This section presents the services and the source of revenue, specific to the Belgian electrical network, that are considered for this study. It also presents how this revenue stream is modelled and optimized. 3.1 Belgian network ancillary services Generally, revenues generated by a PHES installation can be granted as: long term energy exchange contracts, market revenues, day-ahead market (DAM), continuous intraday market, primary (R1), secondary (R2) and tertiary (R3) reserve, black start, loss compensation, tension support and imbalance compensation (Delille 2010). Some of these revenue sources can be cumulated, as DAM and tension support, but others are more complicated to combine. Indeed, energy sold daily will impact the maximum allowable R2 services. A decision must thus be taken in regards of the contribution of every service provided to the network. This optimization depends on various parameters as: market status, ancillary services prices, time of the

year, etc. This study focuses on long term investment

decisions, and the following services, provided to the

Belgian network, are thus considered

• Day-Ahead Market: based on the difference of prices during a day on the DAM. This means to pump when the electricity prices are low and to turbinate when they are high.

• Primary reserve: based on the regulation of the network frequency on an European level. It has to be active in 30 s with a minimum bid of 1 MW. The plant has to be equipped with automatic speed and frequency variation equipments. It is contracted monthly or yearly and paid on basis of the reservated capacity (not on its activation) (Elia 2015).

• Secondary reserve: based on the regulation of the network frequency on a national level. It has to be active in 15 minutes and propose at least 1 MW during 24h. It is contracted daily and paid on the reservation and on the activation through bids. The order of activation is regulated according to the best bids, therefore activation is not mandatory (Elia 2015). For short and mid term operation of the plant, addi

tional revenue streams could be captured. They are

difficult to predict for long term operation and are

therefore not considered in this study.

3.2 Computations and optimization

To estimate the revenues generated from these ser

vices, one needs to know how the plant will be

operated. This can be done two ways: one is to define

simple hypothesis concerning the operation of the

plant (i.e. pumping during the night and turbinate the

night). However, this greatly limits the complexity of

revenue streams. Another method optimizes the oper

ating plan to maximize the revenues. This requires to

state the complete system (potential revenues, operat

ing costs, constraints, control mechanisms, etc.) under an optimization form. This mathematical definition requires a certain number of hypothesis that need to fit the studied objective: estimate the revenues over decades. The formulation does not have to be more accurate than the quality of the input variables (that are difficult to estimate: electricity prices, evolutions of the markets, etc.). Therefore, this optimization is limited to linear formulations for which resolution algorithms exist. The problem is discredited according to time steps.

For every step, the PHES is considered working at stationary conditions. For these computations, results showed that a one hour time step is sufficient to achieve the required result precision.The optimization is based on the operating conditions of a complete month. This allows to identify opportunities spreading on several days (e.g. pumping in the week-end to turbinate in the week) and also to propose a choice between R1, R2 or maximizing the DAM. The PHES plant is thus

modelled as a buffer allowing to store/produce energy Figure 3. Optimized gains for a 4MW and 20Wh plant with different revenues considerations. with a defined efficiency. The selected revenues are computed, based on 2014 electricity prices, as: • DAM: based on 2014 quarter-hourly data have been used as reference for the electricity prices (Belpex 2014). • R1: 200 mHz reservation prices are 46.8 €/MW/h 1 . It has been shown that for each 1 MW reserved for R1-200 mHz, 4 MWh were needed to cope with the worst day activation profile (based on the 2014 frequency measurements on the Belgian network).This is thus taken as the generation limit for the PHES plant studied.

Technically, the PHES plant needs to be equipped with fast reacting electro-mechanical components, or already be running at lower power outputs. • R2 Reservation prices are 22.0 €/MW/h. For small plants, it is difficult to provide 1MW during 24 hours (according to current regulations). Therefore, it is here anticipated that agregators are used to provide R2 to the network. It is then the role of the agregator to combine different storage systems to provide the required regulation. Therefore, only a limited amount of hours need to be dedicated to R2 and a percentage of the revenues are given to the agregator (20%). • Additional costs arising from the revenues are not taken into account at the moment (as taxes). Specific studies on the effect of policy and energy prices variation will be presented in the future The hypothesis chosen in this study are conservative, therefore the results can be considered as a superior limit. Figure 3 summarizes these computations for a 4 MW and 20 MWh PHES plant. It shows that the DAM only revenues option leads to a high usage of the plant, but that corresponding revenues are low. The cash inflow is increased with activation of R2 services. However, the most important financial gain is obtained when also activating the R1. Actually, 1 This value is highly variable, e.g. in the start of 2016 it is of 16.8 MW/h. Table 4. Quarry and mine specifications. ∆H V E P t Q η pt η tot Piping m m 3 MWh MW h m 3 /s / / m / Quarry 35 200000 20 4 5 11 0.93 0,87 150 Mine 250 30000 20 4 5 1,65 0.9 0,84 300

the optimization always prefers to give R1 revenues

the priority in the computations. This result is subject

to change in the future as it will take into account more

specific constraints.

4 APPLICATION TO A QUARRY AND A MINE

4.1 Site definitions

Two unconventional sites are studied in this paper.

They are representative of some of the conditions that

can be found in open air and underground cavities in

Belgium. There characteristics are given in Tab. 3.2. It

shows that the quarry has a large storage volume and a low head, while the mine has a smaller volume and a high head.The water volume in the mine and the quarry are considered at a constant head 2 . Both sites have the same energy levels and installed electro-mechanical power. Both sites are supposed to be used with two electro mechanical equipments: one for pumping and the other for turbinating. Regarding the head and flow rate specifications, the quarry is best equipped with Kaplan turbines while the mine is equipped with a Pelton turbine. Financial, environmental and permit ting characteristics of both sites are considered equal.

The following hypothesis are also taken:

- Only one of the two tanks of the plant needs to be excavated (mine/quarry act as the lower reservoir).

No coating of the artificial tank is required.

- Lifetime of the plant is of 40 years, the discount rate is 4.5%.
- The nearest mid-level tension cabin is located at 1 km of the plant site.
- Pipe length is site dependent. The low head site uses PVC pipes and the high head steel pipes.
- O&M costs are taken as 1.5% of the total investment costs.

4.2 Influence of the site type on the capital investment

At same power and energy levels, the main differences between the sites are technical. Indeed, one of the site

presents a low head and a large volume and the other

has a high head and a low volume (Tab. 4).Total capital

investments are in the range literature values (Tab. 5).

2 Other sites present a higher variation in the head during the

operation that impacts the performance of the turbines, but

this will be studied in future works. Table 5. Total capital investment in €/kW. Site €/kW Quarry 2268 Mine 1038 Literature 500–3600 Figure 4. Costs of different parts compared to the total investment required for each site. It is much smaller for the mine.The differences relative to each part of the site compared to the total investment, and computed with the formulas given in section 2, are shown in Fig. 4. It shows that, relative to the total investment, the excavation and spillways costs are significantly more important for the quarry than the mine. This is mainly due to the smaller excavation volumes and flow rates. However, the piping works and the EM costs take a larger part of the costs in the mine (the absolute value for the electro-mechanical machines are still of the same order of magnitude). It is thus understandable that the difference in topological and hydro characteristics of the two sites impact the investment costs. Supplementary costs might need to be added to the startup of mines. Indeed, some mines access tunnels might be refilled, and new tunnel excavation to install electro-mechanical equipments will then be costly. 4.3 Influence of revenues An example of different revenues generated monthly by a 4 MW and 20 MWh plant are shown in Fig. 3. The revenues for the mine or the quarry change slightly due do the different electro-mechanical machines of different efficiencies. Regarding the hypothesis taken in section 3, the most promising results provided by the

Figure 5. Net Present value with Market/R2/R1 revenues

for the mine and the quarry.

optimization process are given by the combination of

R1, R2 and DAM results. Actually, the revenues pro

vided by R1 are significantly larger and the optimiser prefers to subscribe to R1 services rather than R2 and DAM. The main part of the revenues is thus provided by R1 services. Figure 5 shows the Net Present Value (NPV) of the plant over the years. This value regroups the capital and O&M yearly costs, the revenues and the financial parameters (discount rate, etc.), as given by Eqn. 13

with NPV (y) the net present value in year y, C tot the total capital expenditures, C flow the cash flow repre senting the revenues minus the O&M costs and d the discount rate. This figure shows that with R1 services activated, the plant reaches break-even in 6 and 13 years respectively for the mine and the quarry. The hypothesis on the R1 revenues are currently strong and without this revenue, the monthly cash flow is significantly reduced (as seen on Fig. 3). Therefore the two PHES plants are analysed without this revenue source to assess its impact on the financial viability. The NPV is then reduced during the years, as shown in Fig. 6. The break even is not obtained after 40 years. However, as such hydro plants typically have a longer lifetime, the break even would be close to be achieved after 80 years for the PHES plant located in the mine, but not for the quarry. The study of these two PHES plant in a mine and a quarry with the same power and energy level shows that, with the chosen hypothesis, it seems more inter

esting to pursue the detailed investment study, for a

mine with a high head and a low volume than for a

quarry with a large volume but a low head. Future

works will challenge the hypothesis taken and will also

focus on more specific and realistic cases of mines and

quarries that could be used in Belgium. Figure 6. Net Present value with Market/R2 revenues for the mine and the quarry. 5 CONCLUSIONS Pumped hydro electric storage in unconventional sites in Belgium is discussed due to the lack of classical hydro power sites. This technology is interesting as it is mature, but it is subject to important capital expenditures. A technical-economical study mixing the costs and revenues generated by these sites is mandatory to identify potential sites.This paper presents the cost and revenues breakdown for the installation in a quarry or a mine. A technical-economical analysis is performed on a quarry and a mine providing 4MW, 20MWh services to the network. At same power and energy levels, there are no large differences in the generated revenues (apart resulting from the efficiency). However, investment costs change. Indeed, excavation and spillway costs are higher for the quarry due to the size of the required artificial. With the most optimist revenue streams, the mine would break even in 6 years, while the quarry would need 13 years. With less optimist hypothesis, only the mine would break even after 80 years. Therefore, at this first stage of study with two hypothetical site configurations, it seems more interesting to investigate PHES solution in a mine with an high head and a low volume. However, this needs to be asses in regards of additional costs that might be required for the startup of the mine if access tunnels are refilled. Future works will focus on the study of specific real test cases of mines and quarries available in Belgium. This deeper study will also optimize the use of the site (which is not done here) to find the optimal configuration between tank volume and number of turbines. The revenue stream will also be more finely tuned and the hypothesis assessed in regards of future electricity market trends. ACKNOWLEDGEMENTS This research was supported by the Walloon Region in the framework of the SmartWater project.

Agadis, G., E. Luchinskaya, R. Rothschild, & D. Howard

(2010). The cost of small-scall hydro power production: Impact on the development of existing potential. Renewable Energy 35, 2632–2638.

Andaroodi, M. & Al. (2005). Standardization and design rules for civil engineering works of shp. Technical report, Ecole Polytechnique Fédérale de Laussane, Lausanne, Switzerland.

Belpex (2014). Day ahead market prices. http://www.belpex.be.

Connolly, D. & al (2011). Practical operation strategies for pumped hydroelectric energy storage (phes) utilising electricity price arbitrage. Energy Policy, 39(7), 4189–4196.

Delille, G. (2010). Contribution du stockage à la gestion avance des systèmes électriques, approches organisationnelles et technico-économiques dans les réseaux de distribution. Ph. D. thesis, Ecole Centrale de Llle.

Ding, H., Z. H. & Y. Song (2012). Stochastic optimization of the daily operation of wind farm and pumped-hydrostorage plant. Renewable Energy 48, 571–578.

Elia (2015). Ancillary services. http://www.elia.be/en/products-and-services/ancillary-services. 2015.

Fablbusch, F. (1987, February). Determining diameters for power tunnels ad pressure shafts. Water Power and Dam Construction. Hossain, I. M., S. M. Ferdous, S. Salehin, A. M. Saleque, & T. Jamal (2014). Pump-as-turbine (pat) for small scale power generation: A comparative analysis. Developments in Renewable Energy Technology (ICDRET), 2014 3rd International Conference on IEEE 39(7). Kaldellis, J., D. Vlachou, & G. Korbakis (2005). Technoeconomic evaluation of small hydro power plants in greece: a complete sensitivity analysis. Energy Policy 33(15), 1969–1985. Krajacic, G. & Al. (2013). Analysis of financial mechanisms in support to new pumped hydropower storage projects in croatia. Applied energy 101, 161–171. Ogayar, O., P. Vidal, & J. Hernandez (2009). Analysis of the cost for the refurbishment of small hydropower plants. Renewable Energy 34(1), 2501–2509. Ogayra, O. & P. Vidal (2009). Cost determination of the electro-mechanical equipment of a small hydro-power plant. Renewable Energy 1(34), 6–13. Ores (2015). Tarifs 2015. Technical report, ORES, Belgium. Paish, O. (2002). Micro-hydropower: status and prospects. Proceedings of the Institution of Mechanical Engineers,

Part A: Journal of Power and Energy (216), 31–40. Papantonis, D. (2001). Smal hydro power stations. Technical report, Simeon, Athens. Zhang, Q. F., B. Smith, & W. Zhan (2012). Small hydropower cost reference model. Technical report, Oak Ridge National Laboratory, Tennesse, USA.
Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Numerical investigations of a water vortex hydropower plant implemented as a fish ladder – Part I: The water vortex

N. Lichtenberg

Laboratory of Fluid Dynamics and Technical Flows (LSS), University of Magdeburg "Otto von Guericke",

Magdeburg, Germany

O. Cleynen & D. Thévenin

Laboratory of Fluid Dynamics and Technical Fluidics (LSS), University of Magdeburg "Otto von Guericke",

Magdeburg, Germany

ABSTRACT: The requirements, which ensure good water quality in natural bodies of water in Europe, also involve restoring the ecosystem and therefore free migration of fish upstream and downstream.As a consequence, measures must be taken and structures must be built which allow for this migration. This paper provides a very detailed overview of the resulting geometric and hydraulic specifications for such structures. On the basis of the results of a vortex hydropower plant simulation, the plant's suitability as a fish ladder is underlined, as all requirements in terms of flow velocity are met.

1 INRODUCTION

1.1 Legal requirements

The Water Framework Directive 2000/60/EC requires all European Union Member States to protect and improve the water quality (Europäische Union, 2000). This includes the restoration of the ecosystems in or around a body of water (rivers, lakes and groundwa ter) starting by 2015. Additionally, this requirement includes enabling free migration of fish and other life forms, both upstream and downstream.

1.2 Consequences and possibilities

The largest disruptions to natural fish migration are caused by hydroelectric power plants, in which the kinetic energy of water is converted into electrical energy by means of turbines and generators. To obtain the greatest possible power output of the plant, most installations – which make up a large part of renewable energy production – are dammed, which effectively results in larger pressure drop and thus higher energy potential. Modifying the facilities to comply with the EU directive can only be achieved by disman tling, decommissioning, or substantially modifying the facility. All these processes have something in common: they are always associated to investments and operating costs. The most common compliance modification to dammed power plants is the addition of so-called fish ladders, which provide the aqueous forms of life with ascent and descent paths along its serpentine shape. An alternative to fish ladders which is attracting attention is the vortex power plant: an installation intended to enable fish to swim through without injury, while, in addition, generating electricity by rotating a turbine. When connected to the electricity supply grid, this production then contributes to a return on investment for the plant. Nevertheless, the fish-friendliness of such a powerproducing plant is difficult to assess a priori. The power plant flow characteristics most relevant to river ecosystems will first be reviewed, in order to provide the background by which to assess the performance of a prototype vortex power plant configuration through computational fluid dynamics (CFD) simulations. 2 LOCATION CLASSIFICATION 2.1 Flowing bodies of water Rivers and streams differ wildly in dimensions and inclines. The thus resulting flow rates, widths, depths, and turbulence levels of the water, when uniform over a certain section, therefore provide a habitat for specially-adapted life forms. These regions of uniform characteristics then form a so-called fish region characterized by a specific and representative fish type. The DWA-M 509 technical guideline "Upstream Fishways and Hydraulic Structures Passable by Fish" (DWA, 2014) provides a good overview. For the classification, the most relevant factors are: • Incline • Water flow

Table 1. Classification of dominant sediment (Dick, 1990).

Body of water section Dominant sediment

Stream Upper reaches Stones Middle reaches Stones Lower reaches Coarse gravel

River Upper reaches Fine gravel Middle reaches Sand Lower reaches Silt

• Oxygen content

• Temperature

• Water section

• Sediment

All listed factors are geological factors, which are

determined by nature and to which the flora and fauna

has adapted by evolution.

2.2 Sediment

Along the watercourse, the ground changes according to the prevailing sediment or substrate. In mountain sections this is mostly stones, which further down the course, mostly by erosion influences, turn to gravel, sand, and then silt in the river mouth. A detailed clas sification of the substrate with information on particle size can be found in Table 1 (Dick, 1990). Large bodies of sediment, such as stones, not only produce zones of calmer flow which provide refuge areas for fish; they also produce vortices in their wakes which attract fish attracted to specific flows, such as trout whose natural swimming movement has adapted to the frequency of vortex detachment (Voß et al., 2016).

2.3 Sections of a body of water

In the literature, flowing bodies of water are divided into three broad sections which are directly related to their incline: source, stream, and river. Within this, stream and river are again divided thrice into upper reaches, middle reaches, and lower reaches. In order to ensure international communication.

Table 2 lists the international and Latin names.

2.4 Incline

The incline is among the most relevant criteria (Schw evers et al., 2012) used in the zoning of flowing bodies of water in Germany on the basis of GIS data for

mapping.

Two forms of incline must be distinguished from

one another: The energy gradient I_e , which is based

on the difference in water surface height and the bed

slope I_o , which is based on the difference in ground

(bed) level. If the surface of a natural, free flow (chan

nel) exhibits no significant altitude changes and thus

lies parallel to the ground, it is described as a uniform

channel flow. For this special case, the energy gradient
Table 2. Classification of Inclines (Schwevers et al., 2012). Water section International Incline [%] Source Krenal – Stream Upper reaches Epi-Rhithral >15 Middle reaches Meta-Rhithral 6–15 Lower reaches Hypo-Rhithral 2.5–6 River Upper reaches Epi-Potamal 0.5–2.5 Middle reaches Meta-Potamal <0.5 Lower reaches Hypo-Potamal 0, tidal influence Figure 1. Schematic representation of the incline. I_e equals the corresponding bed slope I_o and is defined as follows: where h_v is the head loss and �x is the difference in the x-direction and simultaneously represents the difference in potential and as such the driving force of flowing water. The topic of the flow resistance and flow properties of low-volume flow bodies is considered in detail by (Jirka & Lang, 2009) River flow conditions are determined not merely by the slope, but also by the available bed cross-section area available. The two factors are related according to the volume flow rate, and a correlation is observed with changes in the fish population in particular geographic regions (Huet, 1949). 2.5 Flow rate/outflow Outflow is described as the entire volume of water per unit time, which leaves a certain catchment area both above and below ground. The flow rate, however, is the flow through the cross-section of surface water. It is a more useful parameter to characterize the water, since the water level may provide important information for ship navigation, etc. The flow rate Q through a facility or a channel portion is constant in a steady flow and thus the multiple

Table 3. Incline types in various sections of a body of

water. Incline [%] for body of water widths of

Section 1m 3 m 15 m 60m 200 m

Epi-/Meta50.025.017.512.5

Rhithral 12.5 7.5 6.0 4.5

Hypo7.56.04.5-0.75

Rhithral 3.0 2.0 1.25

Epi3.02.01.250.75

Potamal 1.0 0.5 0.33 0.25

Meta12.51.00.50.330.25

Potamal 0.0 0.0 0.0 0.0 0.0

HypoTidal influence

Potamal

of the average flow velocity and cross-section area

remains unchanged: Typically, the outflow increases along the course of

rivers, from source to mouth, through the supply of

water from tributaries. As for the cross-section of the

body of water, it also typically expands as the incline

decreases.

2.6 Energy dissipation

Turbulence occurs within the fishways, the degree of

which is expressed as an energy dissipation rate in

hydraulic engineering. It represents the transforma

tion of the energy entering a basin with respect to the

dimensions of the basin. To calculate the energy dissipation according to

(Jirka & Lang, 2009) the specific gravity γ is intro

duced:

wherep is the density and g is acceleration due to grav

ity. Moreover, the definition of the energy loss, h_v, also

comes into play for the special case in which the energy

gradient is equal to the bed slope, $I_e = I_o$ (Figure 1).

Based on this, the energy dissipation is described as

the power loss P_v [W] formed by the product of the

mass flow γQ and energy loss h_v: However, in interpreting the fishways, the spe

cific power density P_D [W/m^3] is usually used (DWA,

2014), which refers to a defined volume of the sys

tem, in order to represent the power against which a

fish must swim while ascending: To ensure exhaustion and injury-free passage for

small and young fish, (BMLFUW, 2012) provides an

overview of design values of individual fish regions. Table 4. Design values for exhaustion and insury-free passage for small and young fish. Height difference between Energy basins D h dissipation P D Section [m] [W/m 3] Epi-Rhithral 0.20 160 Meta-Rhithral 0.18 130 Hypo-Rhithral 0.15 120 Epi-Potamal 0.10–0.13 100 Meta-Potamal 0.08 80 Table 5. Temperature at different water sections. Section Temperature [∘ C] Epi-/Meta-Rhithral 5–10 Hypo-Rhithral 8–14 Epi-Potamal 12–18 Meta-Potamal 16–20 Hypo-Potamal Tidal influence 2.7 Oxygen content The oxygen content in waters decreases from the spring to the water mouth. Indeed, the dominant sediment in the upper regions of the rivers mostly consists of stones and the water section is mainly flat: turbulent flow over large solid bodies promotes wave formation and large turbulent structures through which air is easily ingested. Also, the concentration of dissolved oxygen in river water is higher at low temperatures. Since water temperature decreases generally from source to mouth (Dick, 1990), this generally correlates to decreases in oxygen content. The swimming performance of a fish is highly correlated to the oxygen content of the water. Considerable (up to four-fold) increases in sprinting velocities have been measured by (Richards et al., 2009) when the oxygen content was increased. 2.8 Fish regions All fish species have adapted their form and their behavior to the flow dynamic conditions of a specific habitat. Representative habitats are often selected to classify entire regions with so-called representative fish. In German river regions, the species corresponding to the representative fish are listed in “Upstream Fishways and Hydraulic Structures Passable by Fish” (DWA, 2014). 3 RESTRICTIONS OF A FISH FRIENDLY WEIR 3.1 Fish friendly In a broad sense, a plant can be said to be fish friendly when it enables fish migration with no injury or fatality.

Table 6. Classification of fish regions.

Body of water section Fish region

Stream Upper reaches Upper trout region Middle reaches Lower trout region Lower reaches Grayling region

River Upper reaches Barb region Middle reaches Bream region Lower reaches Ruffe-Flounder-region

Table 7. Geometric maximum requirements for fish region

and largest companion fish. Distance from internals Migration corridor water length depth width

Section [m] [m] [m]

(Fish

species) 3 L Fish 2.5 H Fish 9 D Fish

Epi-/Meta-Rhithral 1.50 0.24 0.45

(brown trout)

Hypo-Rhithral 3.00 0.53 1.08

(Lake trout)

Epi-Potamal 4.80 0.64 2.16

(catfish)

Meta-Potamal 9.00 1.28 3.24

(sturgeon)

Hypo-Potamal Tidal influence

3.2 Geometric requirements

The most significant of the geometrical constraints for fish friendliness are the overall plant transit length, and the gap dimensions (the dimensions of the narrow est passageway through the plant). These requirements need to be adjusted in relation to the dominant fish region. Limit values for height H_{Fish}, width D_{Fish} and length L_{Fish} of the fish are prescribed under the regula tions of “Upstream Fishways and Hydraulic Structures Passable by Fish” (DWA, 2014) and serve as reference dimensions.

These dimensions are chosen not just according to those of the representative fish, but also on companion specie. The largest fish of each fish region is listed in the table below.

According to regulations, the gap dimensions must meet at least 3 times D_{Fish} and its depth must not exceed 2 times H_{Fish}.

3.3 Hydraulic requirements

3.3.1 Rheoactive behavior

The behavior of fish is strongly influenced by the flow. In particular, they spontaneously turn and swim against the flow when they encounter flow velocities exceed ing certain values. The threshold is termed rheoactive

speed and is species-specific; samples corresponding Table 8. Selected rheoactive speeds. Evolutional stage/size Rheoactive speed ish species [m] [m/s] Lake trout 0.12 0.15 Barb adult 0.20 Pike juvenile 0.10 Stickleback juvenile 0.15 Table 9. Minimum and maximum flow rate at the inlet of a fish ladder. Attracting flow velocity at the inlet of a fish ladder Section v min [m/s] v max [m/s] Epi-Rhithral 1.0 2.0 Meta-Rhithral 1.0 1.9 Hypo-Rhithral 1.0 1.7 Epi-Potamal 1.0 1.5 Meta-Potamal 0.8 1.2 to representative species are listed in the table below (DWA, 2014). 3.3.2 Flow attraction In addition to their rheoactive behavior, fish show attractiveness to areas of high local flow velocity, as demonstrated experimentally by (Pavlov, 1989) who correlated capture occurrences behind a dam to outflow rate. The maximum attractiveness was then observed when the speed of the tail water reached 60–80% of the fishes' sprinting velocity. An overview of the minimum flow velocity v min and maximum flow velocity v max is taken from the guide to the construction of fish ladders (AG-FAH, 2011) and displayed below. 3.3.3 Swimming performance of fish The swimming performance of fish varies not only according to species, but also strongly increases along with their body length of the fish. Swimming performance is typically described with three main parameters. The sprint speed is the maximum speed that the fish can attain for a short period. The increased speed is the swimming velocity at which the fish can settle for 200 minutes, typically after a sprint of about 20 seconds (Bainbridge, 1960). It correlates to approximately 5 times the length of the fish per second. The cruising speed is to be regarded as the normal swimming speed of the fish, with which it can move with no noticeable fatigue. 3.3.4 Turbulence A majority of all technical flows are turbulent, and this also applies to the flows in rivers (DWA, 2014). Flow turbulence has a negative effect on the movement of the fish, as it must stabilize itself with its pectoral fins,

Figure 2. Geometric representation of the water vortex

power plant.

Table 10. Modelling overview.

Mesh

Mesh type Trimmed Mesh

Cell number 3 400 000

Mesh adaption Yes

Boundaries

inlet Massflow inlet

outlet Massflow inlet

Massflow ramp 0–850 kg/s

walls no slip

Physical models

Reynolds-Averaged-Navier Stokes Implicit Unsteady

Eulerian Multiphase Volume of Fluid (VOF)

SST K-Omega All y+ Treatment

thereby increasing its hydraulic resistance.This results

in reduced sprint and duration swimming speeds.

4 NUMERICAL INVESTIGATIONS

4.1 Geometry

The pilot plant for the fish-friendly weir in Bühlau,

Germany serves as the base geometry for the numeri

cal investigation. Essentially, it is a cylindrical vortex

hydropower plant. In this first part of the research

work, the plant is studied without a power-producing

rotor; thus, the water is free to flow out. Once it has entered the plant, the water flows

tangentially through the system inlet enforcing a coun

terclockwise circular flow.The water flows out towards

the outlet through a circular opening at the bottom of

the basin.

4.2 Modeling

For the simulation, the software Star CCM+ (company

CD-Adapco, version 10.04.009) was used. In order

to visualize the position and movement of the free

surface, a computationally-intensive VOF model for

Eulerian multiphase simulations was used.All settings

and boundary conditions are listed in Table 10. In the simulation, the rectangular entrance area was supplemented with a ridge in order to reduce recirculation at the entrance and to ease the control of the inlet water height. The system is home to a complex swirl-prone, unsteady flow with a free surface. In order to ease convergence, the flow is ramped up progressively, raising from 0 kg/s (no flow initially) up to 850 kg/s after 10 seconds. In order to optimize computing resources, an automatic mesh refinement at the points of interest such as the free surface and areas of high-velocity flow, was performed. The ensuing grid structure can be seen in Figure 3. 5 RESULTS AND DISCUSSION The simulation was run for a physical time of 90 seconds in order to ensure that the core vortex was fully-established. Figure 4 to Figure 5 display the volumes within the water flow for which the velocities lie within the threshold of interest. Figure 3 presents the velocity magnitude of a plane cutting through the throat of the power plant. The thresholds and color maps displayed in these figures indicate that the main species of fish of interest in this study will likely be able to swim through the device: the attraction velocity is attained in the most significant areas, while the areas where the flow exceeds the expected maximum sustained velocity are constrained. Corridors of acceptable velocity magnitude are observed in the simulation. Moreover, highvelocity areas are mostly located in side regions for example, the locus of the highest velocity (2.7 m/s), is found directly behind the edge of the floor outlet opening. While the attracting flow represents 80% of the critical swimming velocity of a fish, the maximum speed of the Epi-Potamal region is 2.2 m/s and is shown in figure 6. However, Epi-Potamal fish can still traverse this area, as they are able to reach sprint speed for a short period of time. First investigations at the prototype of the company KÄPPLER&PAUSCH in Bühlau, Germany show “that the system can be passed in general. Particularly the passage by smaller individuals and species that are not particularly strong swimmers must be seen as positive here. (…) for 2014, a detailed function inspection over several months under inclusion of fish downward migration and hydraulic parameters is planned" (Proof of Fish Migration – Die Alternative zur Fischtreppe). 6 CONCLUSION AND PROSPECTIVES Although no turbine is involved in this first research work, the complex nature of the flow patterns highlights the need for a CFD analysis in order to assess fish-friendliness of power plants. In this case the simulations are a useful tool for an accurate evaluation of

Figure 3. Velocity profile with mesh size distribution after adaptive meshing at regions of high gradients and free surface.

Figure 4. Velocity threshold above 2.2 m/s representing the area of critical swimming velocity of the Epi-Potamal region

where the the fish-friendly weir is located.

Figure 5. Velocity threshold above 2.0 m/s representing the area of maximum attracting flow velocity of just the Epi-Rhithral

region.

velocities most relevant to facilitating fish migration

of different species. It was found that at its nominal flow rate, the fish

friendly weir considered in this study leads to a flow

which permits passage of species of fish in the regions

in which it is intended to operate (Epi-Potamal).

An immediate topic for further work is investigating

this characteristic when the flow rate and available head are varied both above and below their nominal value. Looking onwards, the addition of the verticalaxis rotor (which gives its purpose to the design) will strongly affect not only the flow distribution throughout the plant, but also bring in new challenges in terms of gap width and specific energy dissipation rate. Investigation of these effects will be presented

Figure 6. Velocity threshold between 0.8 m/s and 1.0 m/s

representing the area of minimum attracting flow velocity for every

fish region.

in upcoming work. Further investigations could also

focus on other complex aspects of the fish-friendliness

of such devices, such as their potential to increase the

oxygen concentration.

ACKNOWLEDGEMENTS

The authors are grateful for the financial support pro

vided by the Fluss-Strom project financed by the

Bundesministerium für Bildung und Forschung (Ger

man Federal Ministry of Education and Research,

BMBF) under the project number 1714.

NOMENCLATURE

Symbols/

Abbreviation Meaning Units

A Cross section m^2

Δh Height difference m between basins

g Acceleration of m/s^2 gravity

I_e Energy gradient %

I_o Bed slope %

EU European Union –

h_v Head loss m

P_v Power loss W

P_D Energy dissipation W/m^3

Q Volumetric flow rate m^3/h

v Flow velocity m/s

v max Maximum flow m/s velocity

v min Minimum flow m/s velocity

V Basin volume m 3

γ Specific mass N/m 3

ρ Density kg/m 3

Performance mapping of ducted free-stream hydropower devices

Olivier Cleynen, Stefan Hoerner & Dominique Thévenin

Lehrstuhl für Strömungsmechanik und Strömungstechnik (LSS)

Otto-von-Guericke-Universität Magdeburg/ISUT, Magdeburg, Germany

ABSTRACT: A series of numerical experiments is conducted to evaluate the effect of ducting on the hydraulic

power production characteristics of a low-impact, low-power, free-surface hydraulic device. A methodology is

presented to identify, for a given duct geometry, the relative duct size which maximizes the device power density.

Results from this calculation-intensive method are compared with a previously-developed theoretical model.

The duct drag behavior observed in experiments differs from that of the model, due to the wide range of flow

patterns observed across size ratios. The accuracy of theoretical predictions may be increased with an improved,

more complex duct scaling process.

1 INTRODUCTION

One area of interest in the search for sustainable energy

production networks is that of low-head hydraulic

devices. Among those, free-surface devices, which do not require damming and thus are able to meet the stringiest of ecological standards, are currently a focus of research and development efforts [7, 11]. One notable weakness of such devices, however, is their typically low power density. Ducts and shrouds, which may be used to increase the power of an exit ing device, or to reduce the required size of its mobile parts for a given power, are often developed to address this issue.

While ducts may possess desirable characteristics such as providing buoyancy or shielding of damage prone parts, they also may also influence output power. When this last influence is used adequately, ducts can present cost reduction opportunities as they are suscep tible to be less expensive than rotating parts. Taking best advantage of these opportunities, as is already being done with constrained-flow installations [8], requires an effective model for quantifying hydraulic power. In this context, it is desirable to develop analytical and methodological tools to assess the influence of ducts on the power of free-surface devices.

2 PROBLEM STATEMENT

In the broadest sense the problem at hand can be stated as follows. In spite of their advantages, ducts inevitably lead to friction losses, a negative effect, so

that a compromise must be reached regarding their

size. Around any particular hydropower device, the use of ducting has two critical effects on operation and performance: • The flow velocities in the device core are typically increased. How is the optimum operating velocity in the device core affected by the duct? • The hydraulic power density of the device core is typically increased. How can this influence be characterized? Does an optimum size always exist for the duct, and which parameters does it depend upon? A theoretical, one-dimensional flow model has been described in a previous publication [10] to provide an answer to these questions. The purpose of the present work is to examine how well this model and the associated methodology measure up to three-dimensional applications in free-surface flow. 3 REVIEW OF THEORETICAL MODEL In previous work [10], the generic characteristics of ducted free-stream hydropower devices have been described with the following parameters. The efficiencies at play, defined in fig. 1, can be applied in series to obtain the output power $\dot{W}$ usable from the maximum available hydraulic power: The power coefficient C_P is computed relative to the device full frontal area A_f :

Figure 1. A conceptual representation of the energy flow

through a machine operating in arbitrary conditions, from

[10]. The ratio of hydraulic power (effectively withdrawn

from the water) to the maximum available mechanical power

in the water is defined as the load efficiency η_{load} . The ratio

of shaft power to hydraulic power is the hydraulic efficiency

$\eta_{hydraulic}$. Lastly, the generation efficiency $\eta_{generator}$ relates

shaft power to useful power (e.g. as electricity). Useful power

is maximized when the three efficiencies are equal to one.

The size of the duct relative to that of the device was evaluated using the size ratio R parameter which compares the frontal area A_f of the device to that of the actuating part, A_A :

The maximum output power of a device had been quantified as a function of device actuator surface area and water surface altitude drop. This involved a static drop coefficient K_{D0} and in particular a duct drag coefficient K_{D2} :

These tools were used to derive an expression for the power density C_P/R; the maxima for power density were attained when the duct ratio reached an optimum value R_{opt} which depended solely on K_{D0} and K_{D2} , and is represented in fig. 2. From fig. 2 it was seen that for any given value of K_{D0} (dictated by the operating environment), increasing values of the drag coefficient increased the optimum size ratio R_{opt} , i.e. the relative size of the actuator that would maximize power density. This figure was intended to provide quantitative, early design phase guidance to maximize the performance of hydropower machines operating in a wide channel.

4 NUMERICAL EXPERIMENT

The present numerical experiment was designed to observe and characterize the effect of ducting around an arbitrary hydropower device operating at the surface of an unobstructed stream, over a wide range of operating conditions. To this effect, a three-dimensional numerical

simulation was conducted reproducing a flat-bed, Figure 2. The optimum actuator size ratio R_{opt} (that will result in maximum hydraulic power density $\frac{1}{R} C_{P\,max.} / C_{P\,max.}|_{R=1}$), as predicted and plotted in [10], represented as a function of the drag coefficient K_{D2} for various values of the static drop coefficient K_{D0} . Figure 3. Cross-sections of the three ducts tested in this study. The power extraction takes place in the thin dotted horizontal line, which is placed perpendicular to the flow. unobstructed river flow. Three geometries, represented in fig. 3, were selected to duct a semicircular hydropower device. Since the focus of the experiment was on the design methodology, rather than the optimization of any particular device, the geometries were selected somewhat arbitrarily among designs expected to allow for reasonably practical applications. Profile A is a foil with medium camber, 5 ∘ angle of attack, and high median thickness intended to ease manufacturing and provide buoyancy; it is similar to that featured in [3]. Profile B is a high-angled, low-thickness diffuser positioned downstream of the power device, as suggested in [1]. Profile C consists of a crudely-shaped, straight-edged pod featuring 30 ∘ angles and is regularly featured in the literature [4, 5, 6, 9]. The hydraulic power extraction mechanism, which was not itself the main focus of the experiment, was simply simulated using a vertical porous surface of uniform isotropic porous viscosity tensor. With this

Figure 4. The profile C represented, from left to right, at size

ratios R = 0,8, 0,5 and 0,3. Each profile geometry was mod

eled with three size ratios and considered at three different

operating speeds.

model, a non-uniform power production within a three

dimensional flow can be implemented without the

complexities and specifics of mobile machine parts

(they will be included in later work). The power extracted from the water was then computed as the surface-integral of the product of horizontal velocity and cross-surface pressure loss, effectively modeling a turbine of hydraulic efficiency equal to 1.

The overall frontal surface A_f of the device was maintained constant, and for each geometry a duct was modeled at three size ratios (R = 0,3, 0,5 and 0,8, fig. 4). In each case, runs were simulated at three different actuator velocities (achieved by adjusting the porous surface viscosity coefficient).The limit case for R = 1, which represents an infinitely thin duct, could be approached using a thin ring-shaped duct yielding R = 0,96 used to ease convergence of the simulation near the edges of the porous body.The main simulation output variables were the hydraulic power extracted from the water, and the duct drag force.

The simulations were carried out using an unsteady implicit volume-of-fluid rans solver using the k-ω sst model on a structured 1.5-million cell grid, using the star-ccm+ software.The grid was refined in areas of interest, with y^+ values ranging from 2 to 250, these values being suitable for the employed turbulence model and near-wall model. Simulation constants were chosen as $u_\infty = 1\ m\ s^{-1}$, $D_{frontal} = 1$ m, in a half-domain 10 m long, 3 m deep and 4 m wide, resulting in duct chord-based Reynolds numbers ranging from $3 \cdot 10^5$

$1.3 \cdot 10^6$. It must be stressed that the general trends across

experiments, and not the precise values for any one

particular case, were the focus of the simulation. Since

identifying three optimum points required no less

than thirty experiments, some compromise regarding

the quality and precision of the cfd simulations was

deemed acceptable in this first, broad mapping of duct

performance characteristics.

5 RESULTS AND DISCUSSION

The power and power density curves of the ten con

figurations simulated are shown in fig. 5. A parabolic

fit has been retained in all figures, because of expec

tations stemming from theoretical studies. Since only

a limited number of simulations could be run within the scope of the study, the curves are too coarsely discretized to locate optimum conditions precisely. Nevertheless, two important trends can be highlighted. First, the power curves are all markedly lowered by the use of ducting. Second, the use of ducting affects, but does not always raise, the power density curves. The first trend is easily explained: the mechanical power extraction process in this experiment is driven by a porosity model in which the pressure loss is directly proportional to the throughput velocity. Consequently, the hydraulic efficiency $\eta_{hydraulic}$ remains equal to one regardless of the flow velocity in the device core. The net effect of the duct, even when $\eta_{load} = 1$, can therefore only be to remove mechanical energy from the flow, and thus reduce the device's power coefficient. Power coefficient increases related to the use of ducts are sometimes reported with some ambiguity in the literature (e.g. [2]). If the device is always properly loaded ($\eta_{load} = 1$), these can only be attributed to increases in hydraulic efficiency (because the specific mechanism used for power extraction benefits from greater velocities or smaller size). The power curves of fig. 5, which are all run at η hydraulic = 1, may perhaps help illustrate this. The second trend—that power density curves are affected, but not necessarily raised—was studied in previous work and a model had been built to describe it quantitatively. In this model, the value of the optimum size ratio (that which yields maximum power density C_P/R) had then been determined as a function of K_{D0} and K_{D2} only. In the present experiment, $K_{D0} = 0$ and one intent was to compare the local optimum for each of the three profiles with the predictions of the one-dimensional model shown in fig. 2. Two obstacles made this comparison difficult. Firstly, at least nine experiments are required to produce any one point on fig. 2; consequently only a very coarse mapping of R-values could be achieved. Secondly, the measured values for K_{D2} showed very wide amplitude variations from one experiment to the other (fig. 6), to such an extent that K_{D2} may not be reasonably considered a constant over the range of parameters studied in this experiment. The high variations in the values of K_{D2} are better understood when the flow patterns around the ducts are observed more closely. A hypothesis underlying the use of K_{D2} is the expectation of proportionality between the duct drag force F_{loss} and the square of actuator velocity u_A, i.e. $F_{loss} \sim A_f u_A^2$, a relation which typically holds true across cases which display high kinematic similarity. In this experiment however, the flow patterns (and not merely the velocity field magnitudes) were strongly affected by both the size ratio and the actuator velocities, as shown in fig. 7. Low actuator velocities and low size ratios, for example, were associated with large separation areas on the outer surface of the ducts, whose influence on the drag force magnitude is not adequately captured by a constant drag coefficient-type relation.

Figure 5. Power curves (left) and power density curves (right) for the three duct geometries tested. The values of C_P are

calculated according to eq. 2. Each graph displays four power curves, one for each size ratio R tested. Measurements from cfd

simulations are shown as scattered points, and curves are proposed as 3rd degree polynomial regression fits. The relation between the duct drag and the flow

parameters can be modeled differently. The duct scal ing process selected in this experiment was such that

the duct cross-section profile remained unchanged; as

such, the wetted surface grew approximately inversely

to R. Scaling the measured drag values accordingly, as shown in fig. 8, suggests that the drag force in this experiment could be approximated by a linear relation of type

Figure 6. Values of K D2 measured in the numerical experi

ments, for each of the three geometries studied. Occurences

at ±1, 5 IQR away from the mean are labled as outliers.

Figure 7. Two extreme flow cases from the study (profile A

at R = 0, 8 and R = 0, 3), shown from exactly the same view

point (duct observed from upstream, below the water surface,

with symmetry plane on the left). Orange ribbons indicate

stagnation and separation lines, and the water free surface is

colored according to non-dimensional velocity. At low oper

ating velocities and low size ratios (right), the stagnation line

is significantly drawn inwards towards the machine core, to

such extent that separation sometimes occurs on the outer lip.

in which u 0 would be a reference zero-drag veloc

ity depending principally on the duct geometry. Three

geometries investigated display differing trends in fig.

8; most notably the reduced drag profile C decreases

sensibly as actuator velocities are increased. Eq. 6 has

some dimensional weaknesses (it requires a propor tionality constant in m 2 s −2) and correlates only mildly with experimental results over the broad scope of flow conditions covered in this investigation. It may never theless provide a coarse, first-order relation with which to improve the applicability of the one-dimensional model described in [10]. In a broader look at the problem stated at the start of this paper, the prediction of an optimum size ratio R opt (for which an accurate model for the duct-generated losses is necessary) is mainly useful in cases where flow around the duct is already fully controlled. In this experience, the range of input flow parameters was extremely wide: the Reynolds number was tripled and the inner-duct velocities doubled across simula tions, provoking large differences in the onset of flow separation and local incidence angles across cases. In

future work, a better scaling process could focus on Figure 8. The reduced drag −F loss /(R − 1) shown as a function of the non-dimensional average actuator velocity, for each of the three duct geometries tested. The duct geometries are sorted by color, and the scatter point symbol size is proportional to the size ratio R. maintaining attachment and improving dynamic similarity across size ratios. In this way, although geometric and kinematic similarity would not be maintained, the behavior of duct drag may be modeled more successfully, resulting in more potent analytical predictions for the optimum size ratio. 6 CONCLUSIONS The experimental investigation of the influence of ducting on the power characteristics of free-surface hydropower devices requires systematic coverage of their power map. Trends identified previously are broadly followed: while increases in power density may be achieved with ducting, no increase in the power coefficient must be expected unless the hydraulic efficiency of the actuating parts directly benefits from the flow condition changes. For any given duct geometry, an optimum duct relative size may exist which maximizes the power density. Following previously-developed theoretical investigations, a methodology is proposed to identify this optimum size ratio experimentally. The need for a large number of individual experiments or simulations (so that for each size ratio the optimum operating velocity may be captured) is highlighted. The simulations conducted in this study, which focused on small hydropower installations, translated into broad variations in Reynolds number, flow regime, and duct local incidence angles. In this type of applications, the losses induced by ducting can no longer be accurately predicted using the concept of duct drag coefficient (K_{D2}). A trend in coarse experimental data suggests that an alternate, linear duct drag prediction model may be used for an analytical first estimate of the optimum size ratio. In future work, a geometrical scaling process may be developed by which the flow regime around the duct is kept unchanged across size ratios. Such a process would narrow down the study on the most practically relevant cases, and could lend itself to more accurate analytical modeling. The experimental costs associated with the development of efficient low-impact hydropower devices could then be reduced.

ACKNOWLEDGEMENTS

The authors are grateful for the financial support provided by the Fluss-Strom project financed by the Bundesministerium für Bildung und Forschung (German Federal Ministry of Education and Research, BMBF) under the project number 1714.

[1] C. J. Lawn. (2003). “Optimization of the power output from ducted turbines”. In: Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy 217.1 (2003), 107–117.

[2] Y. Ohya &T. Karasudani. “A shrouded wind turbine

generating high output power with wind-lens technology". Energies 3.4 (2010), pp. 634–649.

[3] M. Shives & C. Crawford. "Overall efficiency of ducted tidal current turbines". In: OCEANS 2010. IEEE. 2010, pp. 1–6.

[4] W. M. J. Batten and G. U. Müller. "Potential for using the floating body structure to increase the efficiency of a free stream energy converter". In: Proceedings of the Congress-International Association for Hydraulic Research (IAHR). 2011, p. 2364.

[5] W. Batten, F. Weichbrodt, G. Muller, J. Hadler, C. Semlow, M. Hochbaum, S. Dimke, and P. Frohle. "Design and stability of a floating free stream energy converter". In: Proceedings of the Congress-International Association for Hydraulic Research (IAHR). 2011, p. 2372.

[6] J. Hadler and K. Broekel. "Low head hydropower — its design and economic potential". In: Proceedings of theWorld Renewable Energy Congress, 2011, pp. 1464– 1471.

[7] W. M. J. Batten and G. U. Müller. Hylow end of project publishable summary. End of project report. HYLOW, 2012.

[8] P. F. Pelz and M. Metzler. "Optimization of powerspecific investment costs for small hydropower". In: Proceedings of 17th international Seminar on Hydropower Plants. 2012.

[9] F. Weichbrodt, S. Dimke, J. Hadler, and P. Fröhle. Großmaßstäbliche Modellversuche mit einem schwimmenden Energiewandler. In: Dresdner Wasserbauliche Mitteilungen. 2012, pp. 291–199. [10] O. Cleynen, S. Hoerner, and D. Thévenin. "[submission] Characterization of hydraulic power in free-stream installations". In: Energy (2015). Submitted. [11] Wachstumskern Fluss-Strom Plus. Grundlastfähige Energiegewinnung durch ökologisch verträgliche Flusswasserkraftanlagen. German. Oct. 2015. URL: http:// flussstrom.eu/. NOMENCLATURE Symbols A A actuator frontal area [m 2] A f device frontal area [m 2] C P power coefficient (def. eq. 2) [—] F loss drag force exerted on the duct [N] g gravitational acceleration [m s -2] h height to water surface, positive upwards [m] K D0 static drop coefficient (def. eq. 2) [—] K D2 duct drag coefficient (def. eq. 5) [—] m˙ mass flow [kg s -1] R size ratio (def. eq. 3) [—] u velocity [m s -1] W˙ mechanical power gained or lost by the fluid [W] z altitude, positive upwards [m] Greek symbols net difference η efficiency [—] ρ

density [kg m −3] Subscripts A actuator max. maximum opt. optimum loss integral effect of pressure losses ∞ far-field incoming conditions Sign conventions Lengths are measured positive upwards and downstream. Fluid property changes are measured from the point of view of the fluid (i.e. negative values indicate a loss by the fluid, thus a gain for the machine operator). Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

In-situ scale testing of current energy converters in the Sea Scheldt,

Flanders, Belgium

T. Goormans & S. Smets

International Marine and Dredging Consultants nv, Antwerp, Belgium

R.J. Rijke

Water2Energy B.V., Heusden, The Netherlands

J. Vanderveken

AquaScrew bvba, Asse, Belgium

J. Ellison

Blue Energy Canada Inc., Richmond, British Columbia, Canada

R. Notelé

Waterwegen en Zeekanaal nv – Sea Scheldt Division, Antwerp, Belgium

ABSTRACT: Waterwegen en Zeekanaal (W&Z) participated in the European project PRO-TIDE, sponsored by

the Interreg IVB North-West Europe program. The main goal of this project was “to increase the use of renewable

energy by promoting innovative, sustainable and cost effective solutions for tidal energy [. . .], in coastal zones

and estuaries”. Within PRO-TIDE, W&Z organised, with IMDC providing technical assistance, the testing of different current energy converters. This paper describes the set-up of these tests and the main findings. Three different devices were consecutively tested during 4 weeks. Besides characterizing the performance of the devices, the tests aimed at gaining experience in more practical aspects, such as logistics, installation techniques, and maintenance requirements. It appears that further technological development is required to generate profitable energy from the relatively low velocities in the Sea Scheldt. Despite these low velocities, the Scheldt was found to be a challenging environment to install current energy convertors.

1 INTRODUCTION

1.1 PRO-TIDE project

Waterwegen en Zeekanaal nv (W&Z) is a Flemish agency responsible for managing and operating the navigable water courses in the central and western part of Flanders, Belgium. W&Z participated in the Euro pean project PRO-TIDE, funded by the Interreg IVB North-West Europe program (http://www.pro-tide.eu). The main goal of this project was "to increase the use of renewable energy by promoting innovative, sustain able and cost effective solutions for tidal energy [. . .], in coastal zones and estuaries". Besides Belgium also The Netherlands, UK and France were represented.

Within PRO-TIDE, W&Z, with IMDC providing technical assistance, investigated the feasibility of tidal energy harvesting in the Sea Scheldt, both with con

ventional turbines at specific structures along the Sea

Scheldt (i.e. the stretch of the Scheldt influenced by

the tide), and with current energy convertors (CECs).

This paper relates to the second part only. More infor

mation on the feasibility study at the future tidal lock

in Heusden can be found in Goormans et al. (2014). 1.2 Selection of a suitable location The selection of a suitable location for installing the test set-up is described in more detail in Goormans et al. (2015a, b). That investigation started with making an indicative energy density map, based on simulations performed with an existing 2D hydrodynamic model of the Sea Scheldt. On this map, zones with higher energy density could be identified. After considering the requirements of shipping (a test set-up must be situated outside the main fairway) and proximity of infrastructure (for servicing the setup), a ‘long list’ of 28 potential locations along the Sea Scheldt could be drafted (see Figure 1), which was further narrowed down to 8 locations after checking ownership (public vs. private) and the modelled velocity range (wider ranges are more interesting as load for the turbines). Site visits to these locations together with inspection of available bathymetry data led to the deployment of current measurements at 3 locations (Van Troos et al. 2014). From those results, it was decided to install the test set-up at the bridge at Temse (the first road crossing over the Scheldt counting from downstream). This

Figure 1. Overview of the Sea Scheldt, with indication of

the locations on the ‘long list’, and locations with current

measurements.

location showed the highest range of velocities (from

1.20 m/s during ebb up to 1.62 m/s during flood) to

which the turbines would be subjected. Also, it was

outside the fairway and could be serviced by relatively

small vessels. Moreover, the current showed a strong bi-directional behaviour at this location, simplifying the set-up: only two clearly defined directions had to be taken into account.

2 TEST SET-UP

2.1 General

After selecting Temse as location for the test site, it became apparent that, for conducting the tests it might not be opportune to use the bridge pillars for anchor age. Instead, a pontoon of 3 m × 39 m was proposed, oriented with its principal axis along the direction of flow, so as to minimise disturbance. The floating pon toon was moored between two piles, piled in the river bed. The pontoon not only could serve as work plat form during installation of the turbines, but also as storage area for additional components required for the tests, such as a data registration system, a load bank (no grid connection was foreseen because of the temporality of the tests), or spare parts. Two buoys, one upstream and one downstream of the pontoon, were installed to comply with the safety regulations for navigation. The most straightforward way to attach a turbine to a floating pontoon is to make the former floating as well, and anchoring it to the work pontoon. This offers the additional advantage of capturing the high est velocities typically occurring in the upper layers of the water column. Moreover velocity measurement devices can be secured to the floating pontoon as well, so their position relative to the tested turbine assembly remains fixed and is known at all times, contrary to a buoy-mounted device. The velocity measurements were executed with two Recording Current Meters (RCM-9), installed on a steel frame, adjustable in vertical direction. This frame in turn was attached to a cantilever; the horizontal position of the frame on the cantilever was adjustable

Figure 2. Principle sketch of the test set-up (top-down view). Figure 3. Pontoon, cantilever and adjustable frame. The white arrow indicates the position of the vertical frame, to which the RCM-9 is connected. The RCM-9 is below the water surface. as well. Figure 2 shows a top-down schematic view of the test set-up. Figure 3 shows the cantilever and adjustable frame connected to the pontoon. 2.2 Current measurements With the adjustable frame the position of the RCM-9 could be changed depending on the technology being tested. It was positioned along the principle axis of energy extraction, with its depth midway the swept area. The RCM-9 first encountered by the flow during flood is referred to as the ‘downstream’ RCM-9, the other as ‘upstream’. The devices sample the velocity (magnitude and direction) every 10 minutes. The reason for this apparent low time resolution is twofold: on the one hand experience shows that the velocity in the Sea Scheldt hardly varies within 10 minutes, on the other hand this increased battery life, lowering costs for servicing (the pontoon could only be reached by boat). The post processing of the measured data was done in an identical way as those performed for the initial site selection (Van Troos et al. 2014). 2.3 Power measurements Although the practical implementation of the power measurement was different for each technology supplier, they all were required to measure power at the shaft. After all, no grid connection was envisaged.

The shaft power was measured by measuring both the torque and the rotational speed. Reported power figures hence represent 'raw' mechanical power, without losses from electromechanical equipment, such as a gearbox, generator, or voltage converter.

2.4 Calculation of the turbine performance factor

The power P that can be extracted by a CEC is proportionate with the flow velocity cubed:

where η= turbine efficiency; C_p = power coefficient; A = swept surface; ρ= density of the fluid; and v = approach flow velocity. From the actuator disk theory, it can be shown that the power coefficient has an upper limit of 16/27 ≈ 0.593 called the Betz-limit. C_p mainly depends on the rotor design and rotation speed, or, in other words, on the hydrodynamic interaction between the rotor and its surroundings.The efficiency η mainly depends on the electromechanical equipment transforming the rotation of the axis into electricity. In principle η and C_p should be quantified separately, but the in-situ conditions make an accurate determination of C_p very difficult. Therefore, the two factors are deliberately considered as one, called the turbine performance factor. For the sake of simplicity, the notation remains $\eta.C_p$.

The results are processed by performing a least square regression analysis, which results in a curve that could be described using Equation 2:

where K = regression factor. The turbine performance factor hence can be determined by combining Equa tions 1 and 2:

in which a water density of 1000 kg/m 3 was assumed (saline intrusion stops at Schelle, about 7 km down stream of Temse).

2.5 Installation, servicing and demobilisation

Although the tests only concern devices of a modest size, it was important to dispose of reliable logistics. The installation of the test set-up was executed by a contractor specialised in fluvial works. The contractor disposed of a quay along the Sea Scheldt in Burcht, situated approximately 17 km downstream from the bridge (hence towards Antwerp). Turbine parts could be transported over land directly to the quay. The prox imity of the Port of Antwerp to the quay also made the shipping of turbine parts from other parts of the world fairly straightforward.

The turbines were assembled at the quay, and

installed using service vessels of the contractor, equipped with suitable cranes for lifting the turbine assembly in and out of the water. Although the tidal amplitude of the Scheldt can be 5 m and more, draft was never an issue for the installation vessels. However the tides did have an implication on the time at which operations best took place. In order to have the installation at Temse bridge go as smoothly as possible, it was advisable to profit from slack water conditions at site as much as possible. Therefore the departure from the quay had to be planned carefully. 3 TESTED TECHNOLOGIES 3.1 General The aim of W&Z

was to test multiple CECs, not only to obtain broader experience in the practice of harvesting blue energy at tidal rivers, but also to offer the (temporary) facilities for testing to more than one CEC developer. Therefore, after canvassing several parties, each party was invited to submit a technical and financial proposal. Because the project aimed at testing technologies under development, an explicit contribution from the tenderers themselves (in kind or in cash) was required. In the end, three developers were selected. 3.2 Water2Energy Ltd Water2Energy Ltd (W2E) is active in tidal energy since 2007. W2E is mainly involved in the design and construction of the VAWT (Vertical Axis Water Turbine), its floating or fixed support constructions, and prototyping and testing, initially as a side kick of VSH (United Shipyard Heusden) but later as an independent company. The company cooperates with other companies in tidal or blue energy and constructs floaters, models and add-ons. The VAWT technology is applied in many designs, and W2E added a special curve control of variable pitch of the vertical foils/blades (patent pending) to increase the efficiency by 10–15% compared to designs with a fixed pitch, that currently have an hydraulic efficiency of roughly 30–35%. For the test within PRO-TIDE, a VAWT of New Energy Corporation Inc. (NEC) is used, the four-bladed EnCurrent 5 kW (ENC-005) turbine (Fig. 5). The area swept by the rotor is $D \times H = 2.0\ m \times 1.5\ m = 3.0\ m^2$. The curve pitch control of the foils was implemented by W2E as an add-on. The complete test set-up was mounted on modified floaters called “Dragonfly II”, fitted with a special tilting device for the turbine, the modified ENC-005 turbine with stainless steel curve control chamber. Figure 5 depicts the Dragonfly II. 3.3 AquaScrew bvba AquaScrew bvba is a small Belgian company from Flanders, and entered only recently the blue energy market.

Figure 4. VAWT of New Energy Corporation Inc.

Figure 5. Modified floaters “Dragonfly II”, Water2Energy,

and stainless steel curve control chamber (grey disk). AquaScrew develops a horizontal axis turbine, with

an Archimedean screw-like rotor with a progres

sive blade diameter (Fig. 6). The maximum diameter

is 2.7 m. This absence of individual blades should

increase its fish-friendliness. The rotor design used for the tests was the result of CFD simulations and lab

scale testing, executed prior to the PRO-TIDE project. The horizontal orientation requires the rotor to yaw

when a bior multi directional application is required

(as in the Sea Scheldt). For the test in the Scheldt, the AquaScrew was

mounted on a floating device. In a first attempt,

this floating device could not bear the loads, and a

larger structure was designed. The new pontoon was

constructed using Kevlar-reinforced polyester (Fig. 7).

3.4 Blue Energy Canada

Blue Energy Canada (BEC) is a Vancouver, British

Columbia (Canada) based business that is developing

a vertical axis tidal turbine, the VAHT or Vertical Axis

Hydro Turbine, for use in current flows in oceans and
Figure 6. AquaScrew rotor. Figure 7. The AquaScrew being mounted on the floating device (Ostend). rivers. The company was founded to continue work originally conducted at the National Research Council of Canada. BEC participated in an experimental research project at the University of British Columbia in Vancouver, which was instrumental in the proof of concept of the turbine design. The 4-bladed rotor has a self-acting (passive) variable pitch mechanism on each of its blades. The design was developed following tests (prior to the project) which showed the shortcomings of both 3bladed straight and helical turbines with fixed pitch. As an example, Figure 8 shows a 3-bladed model with self-acting variable pitch. For testing BEC constructed a ducted vertical axis turbine with a 4-bladed self-acting variable pitch rotor. The rotor was 2 m in diameter and 1 m in height for an area of 2 m 2 .This was placed in a 3.3 m wide symmetrical floating duct, with a height of 1.35 m. The length of the turbine assembly was 3.4 m, and weighed 1100 kg. Figure 9 shows the turbine, suspended from a crane during installation. The turbine and ducting work equally with the ingoing and outgoing tide. The variable pitch rotor gives improved starting torque and

efficiency over fixed blade vertical axis tidal rotors (Lazauskas & Kirke

Figure 8. Self-acting variable pitch vertical axis 3-bladed rotor model.

Figure 9. VAHT of BEC, installed in its duct. The turbine is suspended from a crane, during installation.

2012), and laboratory tests on this type of ducting have shown an increase in power output (Rawlings 2008) as well as further reductions in shaking and torque ripple (Malipeddi & Chatterjee 2012).

4 RESULTS

4.1 Dragonfly II

4.1.1 Tested periods

The Dragonfly II was in the water from 20 October 2014, and was planned to be retrieved mid-November, but because of delays in the project planning it remained in the water until 11 December 2014 (which allowed for additional testing). Within this period, six sub-periods were identified, suitable for further anal ysis. The other periods generated little useful data due to fine-tuning of the measurement set-up, check-up and adjusting of the turbine rotor, or maintenance. The analysed periods are shown in Table 1. Figure 10 shows, as an example, the measured time series of the flow velocity in the upstream and down stream RCM-9, together with the generated power, for the period 04/11/’14-07/11/’2014. Table 1. Analysed periods in the Dragonfly II campaign. Peak Peak Peak flow flow power ebb flood output Start End m/s m/s W 31/10/’14 01/11/’14 1.23 1.05 2044 04/11/’14 07/11/’14 1.45 1.52 1182 07/11/’14 13/11/’14 1.43 1.59 1954 13/11/’14 19/11/’14 1.31 1.14 552 28/11/’14 03/12/’14 1.40 1.27 910 06/12/’14 08/12/’14 1.48 1.44 1282 Figure 10. Flow velocity and power of the Dragonfly II, from 4 to 7 November 2014. It is clear that the turbine generated power only during flood; this made the test set-up more straightforward. However, with some modifications, the turbine could easily be made bi-directional, because of the vertical axis orientation. The difference in velocity between the upstream and downstream RCM-9 is due to the extraction of energy by the turbine. Even if the turbine is in ‘free ride’, i.e. in the water and rotating but not loaded, there is a velocity difference to be expected, due to the mere presence of the turbine. Finally, it can be seen that the difference between the upstream and downstream RCM-9 is larger during ebb than during flood. This is because the turbine was not positioned in the middle of the work pontoon, but closer to the downstream RCM-9. 4.1.2 Performance Figure 11 shows each data couple for the period 4–7 November 2014. Data couples with a power value below 50 W were not considered in the analysis, since no noticeable flow velocity was present.Also, when the flow was below 0.5 m/s the power values showed incoherent results. This was attributed to the fact that the power measurement set-up was not fitted for measuring at very low flow. These points were not considered either. Besides the data couples, the figure also shows the curve that resulted from the least-square regression

Figure 11. Flow velocity measured by the downstream

RCM-9 vs. measured power of the Dragonfly II, for the period

4–7 November 2014.

Table 2. Turbine performance factors of the Dragonfly II

for each analysed period. η.C p

No. Start End –

1 31/10/’14 01/11/’14 0.291

2 04/11/’14 07/11/’14 0.191

3 07/11/’14 13/11/’14 0.219

4	13/11/’14	19/11/’14	0.270
5	28/11/’14	03/12/’14	0.277
6	06/12/’14	08/12/’14	0.265
all			0.226

analysis. For the depicted curve, the turbine performance factor, calculated using Equation 3, is 0.191. The turbine performance factors for the other periods were analysed as well, the result can be seen in Table 2. The analysis was also repeated by considering all data points as one set, this result is denoted ‘all’. Results are quite consistent, but overall low, compared to more mature technologies, which can reach values up to around 0.40. However it must be noted that these test were the first in-situ tests for the Dragonfly II. Moreover, the number of 0.40 given above relates to technology aimed at harvesting energy from higher flow velocities (2–4.0 m/s) – e.g. offshore conditions – compared to those occurring in most natural rivers, or were derived in a controllable laboratory environment.

4.2 AquaScrew

4.2.1 Tested periods

The first float was installed on 11 December 2014. However, as mentioned above, the floating capacity appeared to be insufficient, and a more robust design was required. This larger assembly was transported from Ostend to the quay in Burcht on 16 March 2015. Unfortunately, on the second day of the test an inci

dent occurred, resulting in a signal loss of the power

measurements at the shaft. An inspection the next day

learned that the vertical suspension collapsed, as indi

cated by the white arrow in Figure 12. It was decided Figure 12. Damaged suspension. The arrow points to a bar that should be vertical. to remove the full assembly from the test site, and have it transported back to the quay in Burcht for further inspection. It then became clear that the rotor was missing. It is unclear what caused the incident and the subsequent damage. Most probably an object collided with the assembly. Given the weight of the rotor (250 kg), the coupling and housing (70 kg) and the floating support structure (140 kg), the object must have had considerable weight and dimensions. 4.2.2 Performance Unfortunately almost no data are available. The data that are available learned that the rotor turned at about 20 rounds per minute, when the flow velocity was about 1 m/s. It was also clear that a flow velocity of 0.5 m/s was required to initiate rotation rotor. 4.3 Vertical axis hydro turbine 4.3.1 Tested periods The assembly of the turbine started on 29 June 2015 on the quay side in Burcht, and lasted for approximately a week. On 9 July 2015 the turbine was installed at Temse, but data collection only began in the beginning of September. Testing was initially planned to include a power electronics package and a capacitor bank. When this was not available in time for testing, it was decided to load the turbine with three 500 W heaters instead, one installed on each phase of the 3-phase power. In the absence of any power electronics and managed load, it was decided to manually change the loading to match the water flow. This necessitated successively turning on each of the three phases as the water flow increased, and turning them back off as it decreased.The switches had to be turned on and off manually. Data collection began on 4 September 2015, with the 500 W loading on each phase from 15 September until 15 October. For the majority of this time the loading was manually changed during the daytime hours and the load was left on the expected peak load for the evening at one, two, or three phases for the evening hours. During the week from 27 September 2015 onwards (spring tide), the load was manually changed on a 24-hour cycle for seven days, with a

Table 3. Analysed periods in the VAHT campaign. Peak Peak

Peak Peak power power flow flow output output ebb flood ebb flood

Start End m/s m/s W W

daytime cycle

14/09/’15 15/09/’15 1.36 1.37 890 1125

15/09/’15 16/09/’15 1.39 1.41 757 821

16/09/’15 17/09/’15 1.41 1.42 1352 1086

17/09/’15 18/09/’15 1.40 1.30 691 632

18/09/’15 19/09/’15 1.36 1.24 662 618

24-hour cycle

27/09/’15 28/19/15 1.43 1.58 726 1310

28/09/’15 29/09/’15 1.47 1.68 787 1369

29/09/’15 30/09/’15 1.49 1.71 809 1456

30/09/’15 01/10/’15 1.49 1.68 838 1399

01/10/’15 02/10/’15 1.46 1.53 709 1382

02/10/’15 03/10/’15 1.45 1.53 586 1049

Figure 13. Flow velocity and power of the VAHT, on 18 September 2015. Two test runs (4 and 5) were performed that day.

daytime cycle resuming the following two weeks. On 15 October 2015 the turbine was removed from site.

Table 3 shows the analysed periods. Each period is about one day long, but can consist of several test runs, depending on the amount of heaters switched on, sam pling frequency of the power measurement, or other processing parameters. Each test run was processed using the parameters valid for that run. It is clear that the week from 27 September, which

was a spring tide, was characterised by higher veloci

ties and, correspondingly, higher peak power. Data processing was similar to that described in

section 4.2.1. As an example, Figure 13 shows the

measured time series of flow velocity and power on

18 September 2015.

The turbine was fit to generate power in both direc

tions. During ebb, the representative velocity to assess

turbine performance was the one measured by the

upstream RCM-9, while during flood the downstream

RCM-9 was used. Figure 14. Flow velocity measured by the downstream (flood) or upstream (ebb) RCM-9 vs. measured power of the VAHT, on 18 September 2015. Table 4. Analysed periods in the VAHT campaign.

No.	Start	End	η.C p ebb	η.C p flood
			–	–
1st period (daytime cycle)				
1	14/09/’15	15/09/’15	0.124	0.150
2	15/09/’15	16/09/’15	0.102	0.120
3	16/09/’15	17/09/’15	0.111	0.127
4	17/09/’15	18/09/’15	0.111	0.107
5	18/09/’15	19/09/’15	0.096	0.100
all			0.107	0.125
2nd period (24-hour cycle)				
6	27/09/’15	28/19/15	0.094	0.124
7	28/09/’15	29/09/’15	0.090	0.124
8	29/09/’15	30/09/’15	0.099	0.133
9	30/09/’15	01/10/’15	0.103	0.123
10	01/10/’15	02/10/’15	0.102	– *
11	02/10/’15	03/10/’15	0.087	0.110
all			0.097	0.124

* The power registration showed values up to 10 kW, which were physically impossible. 4.3.2 Performance The turbine performance factor is calculated similarly as explained in section 4.1.2. Figure 14 shows the data couples taken on 18 September 2015, as well as the regression curve. In this case, the turbine performance factor, calculated using Equation 3, is 0.096 during ebb, and 0.100 during flood. The calculation is done with an area of 3.3 m × 1.35 m = 4.455 m 2 , i.e. the area at the entrance of the duct and not the ‘bare’ swept area of the rotor, because the ducting has to be taken into consideration as well.This way, the reported figures include the efficiency of the ducting as well. The turbine performance factors for the analysed periods are given in Table 4. Again it must be said that the turbine performance

factors are low. On the other hand a tip speed ratio of 2.75 was planned, which would have been controlled through the power electronics loading the turbine (which did not arrive in time). Without the power electronics present, a tip speed ratio of 2.1 was observed. This lower tip speed ratio undoubtedly contributed to the lower turbine performance factor.

4.4 Other knowledge gained

Besides contributing to the technical characterisation and a better understanding of the tested technolo gies, the campaign expanded the knowledge on other aspects of harvesting current energy on rivers such as the Sea Scheldt as well. Accumulation of floating debris is a point of atten tion. Large debris, like the one probably damaging the AquaScrew turbine, is less frequent, but also smaller debris can cause problems. Some parts of the turbine assemblies were ‘hot spots’ for accumulation, causing additional turbulence in the flow. It seemed that rotat ing elements were less sensitive than fixed ones. For example, in December the Dragonfly II turbine was not lifted during approximately 2 weeks, but the foils hardly contained any debris. During the tests, a provi sional, wooden deflection frame could be installed to mitigate this problem, although a more elegant solu tion is recommended so as to minimise any negative effects on the performance. The equipment to convert mechanical to electric power, as well as the power electronics, has a signifi cant impact on the turbine performance.An inadequate

or suboptimal design of the drive train or power elec

tronics lower the performance, so care should be taken

so as to incorporate an efficient and versatile power

management system in the design. There was no systematic monitoring of fish or other

animals, so no general conclusions can be drawn. Nev

ertheless, no signs of collisions or other impacts were

observed when visiting the turbines (the rotation speed

of all turbines was rather limited, ranging from 15 to

45 rounds per minute). Finally, the logistics of even a relatively small pro

totype test such as performed in this study, should

not be underestimated. Constructing a (temporary)

set-up in a river requires specialised equipment. More

over, in tidal rivers, the installation and demobilisation

require careful planning, because of the need for

low velocities. Slack water conditions are preferable.

Therefore, an experienced contractor who knows the

local conditions in the river, is vital for a smooth

execution.

5 CONCLUSIONS AND RECOMMENDATIONS

In the framework of the PRO-TIDE project, three

different current energy converters were tested for 4

weeks each, at Temse bridge in the Sea Scheldt, Flan

ders, Belgium. For this, a floating work pontoon was installed, to which the (also floating) turbine assem

blies could be attached. Current measurements were

performed using RCM-9 devices upstream and down

stream of the turbines, while power was measured at

the axis of the turbines. From the data, turbine
performance coefficients

Coupling of water expansion and production of energy

on public water distribution network

B. Michaux

CILE (Compagnie Intercommunale Liégeoise des Eaux), Liege,
Belgium

C. Piro

Cla-Val, Switzerland

ABSTRACT

CILE produces and supplies drinking water to more

than half a million people in the Greater Liege area.

With a production of 100.000 m 3 of water per day, a

distribution network of more than 3.550 km and about

330 civil engineering structures, the Company is one

of the major player in the water sector in Wallonia

(Belgium). Such an activity would be not possible without

very high professional technical supports such as dig

itized mapping or management of remote monitoring

equipment. Unfortunately, the smallest structures are not
sys

tematically connected to power supplies because of

their situations (wood, countryside). But it is very

important to send/receive information on the quan

tity or the pressure in order to guaranty a continuous

distribution service. After discussion with a local distributor of hydraulic equipment in business with a Swiss Company producing among other things control valves, the project consisting in the recovering of the expansion energy in a small water turbine was born. A flow of 50 l/min with an expansion of 0.6 bar is just enough for the production of energy with a power of 14 W stored in a 12V & 24V DC battery. Then, this one supplies a PLC that communicates with the central supervision system. In the past, the energy was produced with photovoltaic panels in summer and with pre-charged batteries in winter. The new solution allows a reduction of the exploitation costs very quickly because of the high competitive price of the micro-turbine (CLA-VAL e-Power IP), now installed on 3 different networks.

Coastal aspects in the era of global change

Abbott, M.B.; Damsgaard, A., and Rodenhuis, G.S., 1973. System 21, Jupiter. A design system for two-dimensional nearly-horizontal flows. Journal of Hydraulic Research, 1.

Abbott, M.B.; McCowan, A., and Warren, I.R., 1981. Numerical modelling of free-surface flows that are two dimensional in plan. Transport Models for Inland and Coastal Waters. London Academic Press, pp. 222–283.

Andersen, O.H.; Hedegaard, I.B.; Dreigaard, R.; de Giro lamo, P., and Madsen, P., 1988. Model for morphological changes under waves and current. Proc. IAHR Sympo sium on Mathematical Modeling of Sediment Transport in the Coastal Zone (Copenhagen. DHI, Horsholm), pp. 310–319.

Battjes, J.A. and Janssen, J.P.F.M., 1978. Energy loss and set-up due to breaking of random waves. Proceedings of the 16th International Conference of Coastal Engineering (Hamburg, Germany), pp. 569–587.

Brøker, I.; Zyserman, J.; Madsen, E.Ø.; Mangor, K., and Jensen, J., 2007. Morphological modeling: a tool for optimization of coastal structures. Journal of Coastal Research 23(5), 1148–1158. Deigaard, R.; Fredsøe, J., and Hedegaard, I.B., 1986a. Suspended sediment in the surf zone. Journal of Waterway, Port, Coastal, and Ocean Engineering, ASCE 112(1), 115–128. Deigaard, R.; Fredsøe, J., and Hedegaard, I.B., 1986b. Mathematical model for littoral drift. Journal of Waterway, Port, Coastal, and Ocean Engineering,ASCE 112(3), 351–369. De Vriend, H.J.,

1987. 2DH Mathematical Modelling of Morphological Evolution in Coastal Zone. Coastal Engineering, 11, 1–27. Dingemanns, M.W., 1983. Verification of numerical wave equation models with field measurements. Delft Hydraulics Laboratory, Delft Report No. W488, 137 pp. Elfrink, B.; Brøker, I.; Deigaard, R.; Asp Hansen, E., and Justesen, P., 1996. Modelling of 3D sediment transport in the surf zone. Proceedings of the 25th International Conference on Coastal Engineering, (Orlando, FL, USA, ASCE), pp. 3805–3817. Fredsøe, J.; Andersen, O.H., and Silberg, S., 1985. Distribution of suspended sediment in large waves. Journal of Waterway, Port, Coastal, and Ocean Engineering, ASCE 111(6), 1041–1059. Johnson, H.K.; Zyserman, J.A.; Brøker, I.; Mocke, G.; Smit, F., and Finch, D., 2005. Validation of a coastal area morphological model along the Dubai coast. Proceedings of Arabian Coast 2005 (Dubai). Kirby, J.T., 1986. Rational approximations in the parabolic equation method for water waves. Coastal Engineering, 10, 355–378. Lesser, G.R.; Roelvink, J.A.; van Kester, J.A.T.M., and Stelling, G.S., 2004. Development and validation of a three dimensional morphological model. Coastal Engineering, 51, 883–91. Laboratoire d'études maritimes (LEM), 2004, Actualisation de l'étude d'aménagement du port de BOUMERDES Panigrahi, J. K.; Ananth, P. N, and Umesh, P. A., 2009. Coastal morphological modeling to assess the dynamics ofArklow

Bank Ireland. International Journal of Sediment research,

24, 299–314.

Strauss, D.; Mirferendesk, H., and Tomlinson, R., 2007.

Comparison of two wave model for Gold Coast, Aus

tralia. Journal of Coastal Research. Special Issue No. 50,

pp 312–316.

Warren, I.R. and Bach, H.K., 1992. MIKE 21: a modeling sys

tem for estuaries, coastal waters and seas. Environmental

Software, 7(4), 229–240. Watanabe, A., 1987. 3-dimensional numerical model of beach evolution. Proceeding of the Conference on Coastal Sediments'87. ASCE, pp. 802–81. Zyserman, J.A. and Johnson, H.K., 2002. Modelling morphological processes in the vicinity of shore-parallel breakwaters. Coastal Engineering, 45, 261–284. Sustainable Hydraulics in the Era of Global Change – Erpicum et al.

(Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Numerical simulation of scour in front of a breakwater using OpenFoam

N. Karagiannis, T. Karambas & C. Koutitas

Department of Civil Engineering, Aristotle University of Thessaloniki, Thessaloniki, Greece

ABSTRACT: In the present work an innovative numerical approach for the simulation of the wave propagation and the scour at the tow of a vertical breakwater is applied. Using two numerical models, this method seems to yield highly satisfactory results from qualitative point of view.The first numerical model, synthesized on the basis of OpenFoam, is used for the simulation of the wave propagation resulting in the hydrodynamic characteristics of the flow while the second one concerns the sediment transport simulation as well as the scour evolution in front of a breakwater, using the results of the first model. Hence, the combination of the above two models constitutes a holistic numerical approach for the simulation of the sediment transport and the scour in front of a vertical breakwater.

1 INTRODUCTION

Scour in the direct vicinity of coastal structures is considered the most devastating process with regard to their stability, which can lead to the failure of the structures. As far as vertical breakwaters are concerned, they are constructed in the deeper part of the coastal zone and their aim is to protect the coastal area from high waves. The depth in front of a vertical break water is considered constant, hence they do not have to confront all these wave breaking processes inside the surf and swash zones due to the shoaling effect. The incident and reflected waves which generate stand ing waves in front of the vertical breakwater and form the hydrodynamic characteristics of the flow, consti tute the main reason for the scouring/deposition of the seabed.

Scouring at the toe of a vertical breakwater not only can threaten the structural stability but also may lessen its performance (Lee and Mizutani, 2008).

The scouring in front of a vertical breakwater has been studied experimentally over the last decades (Carter et al., 1973, Xie, 1981, Mei, 1989, Sumer and Fredsøe 2000) and numerically as well (Gislason et al., 2000, Bing, 2007, Hajivalie et al., 2008) but extensive research on this subject is currently in progress due to the great importance of scouring with regard to the coastal structures.

An innovative approach has been tried in the present work, where two different numerical models have been used. Firstly, a numerical model, synthesized on the basis of OpenFoam, has been developed and imple mented for the wave propagation in order to investigate the hydrodynamic characteristics of the flow. Conse quently, a second model, developed in FORTRAN, has been implemented with the aim of the scouring investigation in front of the vertical breakwater, using the hydrodynamic results of the first model. The aforementioned former numerical model was created with the open source toolbox OpenFoam and the additional toolbox waves2Foam (Jacobsen et al. 2012). The RANS (Reynolds averaged Navier Stokes) equations have been solved simultaneously with the transport equations of the turbulence model k-ω SST, which, after extended investigation, was found the most suitable for free surface cases, and the VOF (Volume of Fluid) method ones (Hirt and Nichols, 1981). The second numerical model has been developed in FORTRAN, estimating the sheet flow sediment transport rates with the Camenen and Larson (2007) transport rate formula, as well as the bed load and suspended load over ripples (Karambas, 2012). Suspended sediment transport rate is incorporated by solving the depth-integrated transport equation for suspended sediment (Karambas, 2012). After that, the conservation equation of the sediment mass is applied for several time steps and the scour is computed. The method used in this work is iterative. Specifically, the first model is applied resulting in hydrodynamic characteristics of the flow, which are used from the second model to calculate the sediment transport rates and the erosion/accretion as well. The new depth, which is the result of the second model, is used by the first model and the process is repeated until convergence of the results, namely morphological equilibrium of the seabed.

2 GOVERNING EQUATIONS OF THE HYDRODYNAMIC MODEL OPENFOAM

2.1 Brief description of the model

This section provides the governing equations of the mathematical model which is synthesized on the basis of OpenFoam and solves the Navier – Stokes equations numerically using the Finite Volume Method. The InterFoam solver is used in this work, suitable for free surface flows. Free surface is tracked by the VOF method. Moreover, the k-ω SST turbulence model is used for the simulation of the turbulence effects.

2.2 Continuity and RANS equations

The mathematical model solves the continuity equa

tion, which is as follows:

in conjuction with the Reynolds Averaged Navier

Stokes (RANS) equations, which are the following

ones:

where U is the velocity, ρ is the density, g is the gravity

acceleration, p is the pressure, μ is the dynamic vis

cosity, and −ρ u ′ i u ′ j is the Reynolds stress tensor
which

is equal to the following expression:

where μ t is turbulent viscosity coefficient, k is the tur

bulence kinetic energy and δ ij is the Kronecker delta.

The last term in equation (1) represents the surface

tension effect.

2.3 Volume of fluid (VOF) equation

Volume of fluid method is provided by OpenFoam

for the “tracking” of the free surface during free sur

face simulations. Using this method, every free surface

computational cell is divided in two parts, one which

represents the air volume and the other is equal to

water volume. The calculation of the water-air por

tion in every cell is possible with the help of the scalar

quantity γ , with its value fluctuating between 0 and

1. When the cell is full of water, γ is equal to 1 and

when the cell is full of air, γ is equal to 0, while it takes intermediate value (between 0 and 1) when the

cell contains both water and air. The γ value is calcu

lated with the following equation for every free surface

computational cell:

where U_r is a relative velocity.

Using γ , quantities like density or viscosity can be

calculated for every free surface cell as follows: 2.4 Turbulence modeling The transport equations for the k-ω SST model are as follows (Menter F. R., 1993-1994): where k is the turbulent kinetic energy, ω is the dissipation rate, ρ the density, U the velocity field, μ the viscosity and μ_t is turbulent viscosity coefficient. The rest coefficients are given in the literature. 2.5 Wave generation and absorption Cnoidal waves are implemented at the left boundary of the model domain, using the additional toolbox waves2Foam (Jacobsen et al. 2012). The equation implemented, describing the above waves at the inlet, is as follows: where H is the wave height, L is the wavelength, T is the wave period and η_2 is the trough elevation. Further Cn is one of the Jacobi elliptic functions and K(m) is the complete elliptic integral of the first kind. Moreover, sponge layers from waves2Foam libraries are implemented at the left and right end of the computational domain in order to avoid wave reflection which would affect the numerical results. The wave attenuation at the sponge layers is described by the following equation: where the α_R is used in the following equation: where ϕ may be the velocity or the quantity γ from the VOF equation. 3 GOVERNING EQUATIONS OF THE SEDIMENT TRANSPORT MODEL The mode of sediment movement on the coast is usually divided into bed load, suspended load and sheet flow transport. Different model concepts are being presently used for the prediction of each one, which range from empirical transport formulas to more sophisticated bottom boundary layer models.

In the present work, the bed load transport (q_{sb}) is

estimated with a quasi-steady, semi-empirical formu

lation, developed by Camenen, and Larson, (2007):

where the subscripts w and n correspond, respectively,

to the wave direction and the direction normal to

the wave direction, $s(=\rho_s/\rho)$ is the relative density between sediment (ρ_s) and water (ρ), g the acceleration due to gravity, d_{50} the median grain size, a_w, a_n and b are empirical coefficients (Camenen and Larson 2007), $\theta_{cw,m}$ and θ_{cw} the mean and maximum Shields parameters due to wave-current interaction, θ_{cn} the current-related Shields parameter in the direction normal to the wave direction, and θ_{cr} the critical Shields parameter for the inception of transport. The net Shields parameter $\theta_{cw,net}$ in eq. 5 is given by:
where $\theta_{cw,on}$ and $\theta_{cw,off}$ are the mean values of the instantaneous Shields parameter over the two half periods T_{wc} and T_{wt} ($T_w = T_{wc} + T_{wt}$, in which T_w is the wave period and $\alpha_{pl,b}$ a coefficient for the phase lag effects (Camenen and Larson 2007). The Shields parameter is defined by
with U_{cw} being the wave and current velocity, f_{cw} the friction coefficient taking into account wave and current interaction and the subscript j should be replaced either by onshore or offshore.
Phase-lag effects in the sheet flow layer were included through the coefficient a_{pl} (Camenen and Larson, 2007) with:
where ν is the kinematic viscosity of water, $U_{w,crsf}$ the critical velocity for the inception of sheet flow, U_w is

the wave orbital velocity amplitude, W_s the sediment

fall speed and the subscript j should be replaced either

by onshore or offshore.

The suspended sediment load (q_{ss}) may be obtained

from (Camenen and Larson 2007): where c_R is the reference concentration at the bottom, ε the sediment diffusivity, and $U_{cw,net}$, the net mean current. The bed reference concentration is written as follows based on the analysis of a large data set on sediment concentration profiles (Camenen and Larson, 2007): where d_* is the dimensionless grain size: The sediment diffusivity was related to the energy dissipation from wave breaking according to Karambas and Koutitas (2002). Phase-lag effect in the suspended concentration due to ripples, is also incorporated according to Camenen and Larson (2007). The nearshore morphological changes are calculated by solving the conservation of sediment transport equation (Leont'yev, 1996): where z_b is the local bottom elevation and q_x (=$q_{s,x}$ + $q_{b,x}$), q_y (=$q_{s,y}$ + $q_{b,y}$)are the volumetric sediment transport rates in x and y horizontal directions respectively. 4 GEOMETRY OF THE MODEL – MESH GENERATION The geometry of the model was chosen in such a way that all model distances are based on realistic ones. There is a 550 m long and 10 m high wave channel with a 10 × 7 m vertical breakwater at 500 m from the left side of the channel. The still water depth in the constant-depth region of the wave channel is 5 m. The 2-D numerical model geometry is depicted below (Figure 1). The mesh was generated with standard OpenFoam tools, specifically with blockMesh and snappyHexMesh. First of all, the basic structured mesh with same-sized computational cells was created with blockMesh and then snappyHexMesh created the vertical breakwater refining the appropriate cells. The discretisation used is x = 1 m and �z = 0.1 m. Appropriate boundary conditions were implemented. Figure 1. Model geometry.

5 NUMERICAL EXPERIMENTS

5.1 Wave characteristics

One cnoidal wave has been implemented into the com

putational domain of the OpenFoam platform in order

for the hydrodynamic characteristics of the flow to be obtained. The wave height is 1 m and the wave period is 6 sec, imposed on the left boundary of the domain through a suitable boundary condition of the additional waves2Foam (Jacobsen et al. 2012) toolbox.

5.2 Hydrodynamic results

It has been already mentioned that the method applied in this work is repetitive. The hydrodynamic model is run for a specific depth, and after that the sediment transport model is run and the new depth is arisen. The latter is used from the hydrodynamic model for the next run until convergence, namely until the new depth arisen from the second model is almost identical with the one from the previous run.

Instantaneous velocities along x and z axis and velocity vectors, time-dependent velocities at the seabed and surface elevation with respect to distance along the x axis are obtained from the OpenFoam model after it was run for 30 wave periods. Veloc ity vectors and surface elevations for specific time instants, during the last wave period (174 s–180 s) the model was run, are presented below.

Figures 2–15 refer to the first simulation where the depth is horizontal. Figure 2 depicts the velocity

Figure 2. Instantaneous velocity vectors at t = 174 s.

Figure 3. Surface elevation at t = 174 s.

Figure 4. Instantaneous velocity vectors at t = 175 s. Figure 5. Surface elevation at t = 175 s. Figure 6. Instantaneous velocity vectors at t = 176 s. Figure 7. Surface elevation at t = 176 s. Figure 8. Instantaneous velocity vectors at t = 177 s. Figure 9. Surface elevation at t = 177 s. Figure 10. Instantaneous velocity vectors at t = 178 s.

vectors at time t = 174 s from the beginning of simu

lation while figure 3 shows the surface elevation at

the same time. It can be clearly seen that a stand

ing wave is formed in front of the vertical breakwater

with the node being at approximately 490 m, where the

velocity vector is totally horizontal and the antinode

at approximately 481 m where the velocity vector is

totally vertical.

Figures 4–5 depict the instantaneous velocity field

and surface elevation for t = 175 s, where can be seen

the evolution of the standing wave which tends to

return to its initial equilibrium state since the velocity

vectors head to the opposite direction and the ampli

tude of the standing wave is reduced. This state is

Figure 11. Surface elevation at t = 178 s.

Figure 12. Instantaneous velocity vectors at t = 179 s.

Figure 13. Surface elevation at t = 179 s.

Figure 14. Instantaneous velocity vectors at t = 180 s.

Figure 15. Surface elevation at t = 180 s. achieved at t = 176 s (Fig. 6–7), where the surface elevation is practically zero with respect to the still water level. Figures 8–15 show the evolution of the standing wave until the t = 180 s time step (Fig. 14–15), which is the start of

the next wave period, identical one with the state of the t = 176 s time step. It can be derived that the OpenFoam numerical model corresponds very well with this kind of problems, namely the standing wave formation due to the wave reflection from the vertical breakwater.The same kind of results can be produced for every run with a different bathymetry after the run of the sediment transport numerical model. 5.3 Sediment transport – scouring results After obtaining the hydrodynamic results from the OpenFoam model, the sediment transport model runs using the near bottom horizontal velocity time series from the former model. The new bathymetry is obtained and compared to the previous one. This process is repeated until the new bathymetry coincides with the previous one and the maximum depth of scour in front of the vertical breakwater remains practically constant after every run of the model. It must be mentioned that the numerical experiment concerns medium sand with average grain size $d_{50} = 0.3$ mm. Fourteen runs were required until convergence comes and the scour takes its final value. It was the last 3 times that the respective bathymetries had about the same morphology. The seabed morphology evolution with respect to time, as arose from the combination of the aforementioned two models is presented below. It can be derived from the first run that the scour has been formed in front of the breakwater and the hole depth is around 0.10 m. Moreover, it seems that a bar is formed at approximately 490 m from the start of the channel, where the node of the standing wave is.A little scour is also depicted in Figure 16 at approximately 480 m from the left boundary, where the antinode of the standing wave is. So, the seabed morphology follows the hydrodynamic state of the domain. Figure 17 depicts the evolution of the seabed morphology after the 3rd run, comparing the 1st and the 3rd run in order to show that the scour goes deeper in the seabed approaching a depth of approximately 0.20 m, namely two times the depth of the 1st run. Figure 16. Seabed morphology after the 1st run.

Figure 17. Seabed morphology after the 3rd run.

Figure 18. Seabed morphology after the 5th run.

Figure 19. Seabed morphology after the 7th run.

The depth of the scour is increasing rapidly as it can

be observed in Figure 18. It takes a value of approxi

mately 0.35 m, which is 3.5 times the depth of the 1st run and 1.5 times the depth of the 3rd run. Tiny dif ferences in the formation of the rest of the seabed in accordance with the hydrodynamic characteristics of the standing wave are depicted in Figure 18 as well.

The scour depth has already reached the value of 0.5 m after the 7th run, as can be shown in Figure 19. After seven runs, the pace at which the scour is increas ing is pretty much the same between the consequent runs and it does not seem to get reduced in order for the bathymetry to reach an equilibrium state. Furthermore, the bar formed at about 490 m is increasing a little, as well as the scour at about 480 m from the left boundary. A second small bar seems to be created, probably due to the different hydrodynamic characteristics between the cases with different bathymetry.

Figure 20 depicts the 9th run in comparison with the 7th one and it seems that the pace at which the scour depth increases has dropped. The bed seems to take a more consistent formation.

Figures 21 and 22 depicts the formation of the

seabed until 12th run, where it seems that the pace Figure 20. Seabed morphology after the 9th run. Figure 21. Seabed morphology after the 11th run. Figure 22. Seabed morphology after the 12th run. Figure 23. Seabed morphology after the last run. at which the scour increases has been reduced significantly. Finally, the seabed has been formed according to the dominating hydrodynamic state, as can be observed in Figure 23 where the last three runs seem identical and the scour depth has reached its final value of about 0.75m. It seems logical in terms of the value of the wave height, which is 1m.Also some oscillations of the seabed morphology seem logical due to the standing wave.

6 CONCLUSIONS

Wave propagation towards an impermeable vertical breakwater has been implemented in the present work and the formation of the standing wave due to the reflection from the structure has been studied as well as the scouring at the tow of the breakwater. An innovative approach has been tried, applying two different models repeatedly. The first model is a hydrodynamic one which has been developed on the OpenFoam platform and the second one is a sediment transport model.

Some important observations can be inferred briefly from the present simulation:

- A realistic case has been chosen in order for the scour to be calculated. The final depth (0.75 m) of the scour is considered naturally occurring with respect to the given wave height of 1 m, as expected.
- The hydrodynamic OpenFoam model seems to behave very well with respect to the standing wave formation and evolution, as analytically presented in paragraph 5.2 during one wave period. There are no experimental results to be compare with the numerical ones, but it can be clearly seen the evolution of the standing wave from qualitative point of view,

according to the literature.

- Velocity field and surface elevation numerical results are in consistency, as can be observed at the specific points of nodes and antinodes and their general shape as well.
- The combination of the hydrodynamic and the sedi ment transport model seems to yield realistic results, as can be derived from the formation of the seabed during the repeated process which applied in the present work.
- The scour increases rapidly at every new run with a new bathymetry at the beginning and until it reaches the 90% of its final depth. In addition, it seems that the pace at which the scour increases remains constant until the time step convergence is reached.
- The seabed morphology follows the hydrodynamic state of the domain, forming a bar at the node and scour at the antinode.
- The last 3 runs of the model seem to yield identical results in terms of the bathymetry and the scour for mation. Some little oscillations on the seabed show the impact of the standing wave on the seabed.
- The present numerical approach seems to behave very well from qualitative point of view, hence a comparison with experimental results will be the

Application of an unstructured grid tidal model in the Belgian Continental Shelf

Biniyam B. Sishah & Margaret Chen

Department of Hydrology and Hydraulic Engineering, Vrije Universiteit Brussel, Brussels, Belgium

Olivier Gourgue

Department of Hydrology and Hydraulic Engineering, Vrije Universiteit Brussel, Brussels, Belgium

Flanders Hydraulics Research, Flemish Government, Antwerp, Belgium

ABSTRACT: This paper presents efforts to simulate tides in the Belgian Continental Shelf (BCS) using the numerical model TELEMAC-2D. Simulations are performed during the calm period (in terms of wind) of July 2013, in order to reduce the impact of meteorological conditions.

1 INTRODUCTION

The vertical rise and fall of the water surface in oceans is referred to as tides. It is caused by astronomical forcing due to phases of the Moon and the Sun around the Earth (Parker, 2007). Tides, in combination with strong winds and low pressure zones play a crucial role in the formation of storm surges (McInnes et al., 2003). If a particular high surge occurs during a tidal maximum, their cumulated effect might cause catas trophic floods in coastal locations. This phenomenon is one of the major natural threats for the Belgian coastal areas and the region surrounding the Scheldt basin. It is also true that the effects of storm surges had been observed increasing through the 20th century and will certainly accelerate in the 21st century due to climate change induced global sea level rises (Nicholls & Cazenave, 2010). Hence, developing a clear understanding about coastal sea water level fluctuations in the Belgian Continental Shelf (BCS) area using a well calibrated numerical model is important (Ntegeka and et al, 2012).

The area of interest, BCS lays in the co-ordinates $51^{\circ} 5' 27''$ N and $51^{\circ} 52' 12''$ N latitudes and $2^{\circ} 14' 17''$ E and $3^{\circ} 21' 53''$ E longitudes. It covers a total surface area of 2695 km^2 and has an estimated offshore length of 63 km (computations were made from model grid).

This location is mainly dominated by Semi-diurnal tides (Verfaillie et al., 2006).

The dominating force in tidal oscillation is astronomic force. It is responsible for much of the tidal oscillation in different locations of the Earth. These oscillations have low amplitudes in deep waters. However, in coastal locations amplitudes become larger due to increased bottom friction and presence of tidal flats (Giardino, 2008). Numerous researches conducted in the field of coastal science have greatly revealed these mechanisms accurately (Woth et al., 2006). Hence, nowadays, more focus is provided upon increasing performance of numerical models to predict costal sea level fluctuations. Among the different flow-modelling computational choices, Finite Elements Method (FEM) was used in this research, due to its capability of defining irregular coastal boundaries and flexibility in discretization of model domain. Studies made by (Walters, 2005; Jones & Davies, 2007, 2008, 2011) have stated out that the ability of this scheme to describe multi-scale processes in one grid with variable mesh densities had revealed the small-scale variability that were not found when using other numerical schemes. In the framework of this research, TELEMAC-2D v7p0r1 was setup covering the entire northwestern European continental shelf with the special focus of calibrating a tidal model that will serve as a basis for future storm surge studies in the Belgium Continental Shelf (BCS) area. 2 MATERIALS AND METHOD 2.1 Model setup TELEMAC-2D is a free surface flow model. This software was developed by the Laboratoire National d'Hydraulique et Environnement (LNHE), a research department of the French Electricity Board (EDFDRD). It has high capabilities for modeling the hydrodynamics of free-surface flows. It was developed for over 20 years and now is an open source software. This two-dimensional finite elements model solves the shallow water equations with both Cartesian and Spherical coordinates, (Hervouet 2000).

In this study, unstructured grid of variable mesh densities, covering the entire Northwestern European continental shelf area and constructed in Cartesian coordinates were used in model. In model setup, the meteorological forces were not considered and a con stant value of the Coriolis coefficient suited for the BCS was used. In terms of boundary conditions, water depths and velocities were imposed at the shelf break using the regional OTIS tidal solution on the European continental shelf, providing amplitudes and phases of 11 tidal harmonic components with a spatial resolu tion of 2 arc-minutes (Egbert et al. 2010). The same data set was used for setting the initial conditions. For large computational domains such as the one considered in this model, the use of spherical co ordinates and a space varying Coriolis coefficient are important. However, that was not possible with the ver sion of TELEMAC-2D used for this study (v7p0r1), due to a bug, now corrected in the current version (v7p1).

2.2 Simulation period

Simulation period was chosen based on wind influ ences in the BCS. A simulation period without the influence of wind (Calm wind period) is as much tried to be considered in this study. This is, since the influ ence of wind forces on measurement station’s reading of tide levels would be insignificant, one can directly assess performance of the tidal model by comparing time series tide levels of model and a measurement station.

Wind data for this study were obtained from the European Center for Medium-Range Weather Fore casts (ECMWF). They are of a radar data for 2013 with temporal and spatial resolutions of 3 hours and 5 arc-minutes, respectively.

Selection of wind period was made by first sub dividing the data set into twelve independent periods, each running from the first to the last day of a month.

Then, Wind Velocity norms averaged in time (month)

and in space (BCS) were computed, of which the month

with the lowest value (calm period) is considered for

analysis. Equations used are as follows,

where w i s is the average wind velocity in space and time

over the BCS in m/s, w i is the average wind velocity

norm of a single period in m/s, W t x and W t y are eastward

and northward velocity at a node in m/s, respectively.

Variables N and t represents number of nodes in the

BCS and time in seconds, respectively. A bar plot dia

gram is provided in Figure 1. Results show that July

is the month with the lowest value and therefore the

calm period. Figure 1. Wind Velocity norms averaged in time (month) and space (BCS) for the year 2013. 2.3 Performance indicators Different statistical methods were used to assess model's performance. The harmonic relative error was given high priority in this study, as it was recommended by (de Brye et al. 2010; Jones and Davies 1996)for tidal studies. Error values obtained with this method are not exact. Hence, results should be interpreted as accurate estimates. In developing its equation, the residual error components e 1 i and e 2 i in m are computed initially at specific nodes: where A o i and A m i represent the ith constituent amplitudes of reference data set and model at a given node, respectively ε o i and ε m i represent phases of reference data set and model, respectively.All constituents available in the reference data set (M 2 , S 2 , N 2 , O 2 , Q 1 , K 1 , MN 4 and M 4) were considered during analysis. T-tide Matlab toolbox (Pawlowicz et al., 2002) was used in the extraction of tidal constituent information from model results. Then the Relative Cumulative Harmonic Error at a node (e s) is given by; where, k stands for the eight tidal constituents considered in the study. The Average Relative Cumulative Harmonic Error over the BCS e s,total is given by the following equation: where, M represents, the total number of observation points considered in the analysis. Another performance indicator used for this study was the Nash Sutcliff efficiency (NSE). It was used to

asses models performance by comparing time series

of model and measurement station's tide levels.

where O_i is the observed tide level; P_i is the predicted

tide level and O is the mean value for the observed

tide levels. The coefficient has values ranging from

$-\infty$ to one. A value of one corresponds to perfect

match between model and observation data set. Mod

els with the coefficient between 0.5 and 1 represent

good performances.

Correlation coefficient (R^2) is the other perfor

mance indicator used in this study. Degree of corre

lation between temporal tide level results of model

and measurement stations records of similar locations

were compared along the shores of BCS. The compar

ison were made at five locations were measurement

stations data were available (Figure 2).

An R^2 of 1 indicates perfect fit between compered

data sets while an R^2 of 0 indicates no fit. The equation

is provided below:

where σ_p and σ_o represents standard deviations of tide

levels of model and measurement stations, respec

tively, Z represents total number time steps, P is the

mean value of model tide levels. The correlation coef ficient is computed for 5 measurement stations that

are indicated on figure 2.

Mean values expressed in equations (7) and (8),

and standard deviations expressed in equations (8) are

given by the following equations:

where, x_i must be replaced either with O_i or P_i in the

above two equations.

Absolute Mean Error (Abs.Error) was also used to

compute the average bias (in meters) for temporal tide

levels of model with respect to actual values of the

variable at the five measurement stations indicated in

section 2. This performance indicator can yield results

between 0 and ∞; a lower value indicates good model

performance. The equation is indicated below: Figure 2. Location of observation stations. 1) Bol van Heist, 2) Nieuwpoort, 3) Oostende, 4) Zebrugge Leopold ll-Dam and 5) Bol Van Knokke. Performance indicators expressed above do not provide enough statistical evidence to assert a holistic model performance as they express the goodness of fit within a single value. Hence, using a more advanced graphical method would fill in the above gaps (Willems, 2009). Accordingly, time series of high/peak tide levels collected from model and a measurement station considering an independency period of 43,200 sec or 12 Hr were plotted together and goodness of fit was used to evaluate model performance. The independency period yielding good result was closer to one period of M_2 constituent. This is because the constituent is dominant in the location and for that matter controls peaking of tides in the BCS (Fettweis and Eynde, 2003; Baeye, 2012; Van Lancker, et al, 2012). 2.4 Reference data set and measurement stations Five measurement stations were considered for performing temporal tide level comparisons with model results.They were obtained from (Monitoring Network Flemish Banks, 2015). Another important data set, the OTIS regional tidal solutions database for the European continental shelf was used as a reference data set to perform nodal comparisons with model results. It consists validated amplitudes and phases of 11 tidal harmonic constants(for tides, U and V velocity constituents) with a spatial resolution of 2 arc-minutes (Egbert et al. 2010). 3 RESULT AND DISCUSSION 3.1 Calibration for bottom friction Bottom friction plays a significant role in adjusting tidal elevations and currents near shallow coastal zones. It is often expressed by empirical equations that further requires calibration procedures (Fernandes et al., 2002). TELEMAC-2D model provides different choices to formulate the bottom friction of which the

Figure 3. Sensitivity analysis of Manning's coefficient for

Velocity components (u and v) and tidal constituents.

Figure 4. Distribution of the Relative Cumulative Har

monic Error for tidal constituents (Manning coefficient is

0.024 s/m 1/3) in the BCS region.

Manning's equation was used in this study for its inclu

sivity of bottom roughness and depth of flow during

computations (Hervouet, 2007). Practical ranges of the

Manning coefficient were tested in model and results

of tidal constituents obtained were cross-matched with

values of the constituents in the BCS region obtained

from a reference data set (OTIS tidal solution of the

European continental shelf). In doing so, the Aver

age Relative Cumulative Harmonic Error expressed

by equation (6) was used. The optimum value of the

Manning's coefficient was found to be 0.024 s/m 1/3 .

Similar procedures were followed to determine opti

mum value of the Manning's coefficient for u and v

velocity constituents. Eventually, for v-velocity opti mum Manning's coefficients was 0.024 s/m 1/3 and model performance was quite good. However, in case of u-velocity model performance was not satisfactory. This might be associated with the lower performance of u-velocity constituents observed at the open bound ary. The findings above are reported in Figure 3 and 4.

In Figure 4, a node-to-node spatial comparison of model's and reference data set's tidal constituents have Figure 5. Comparison between model and reference data set tidal constituents information in the BCS. (Red: M 2 , Green: S 2 and Blue: M 4 constituents). been made using the Relative Cumulative Harmonic Error (e s) method presented by equation 5 and a contour plot of the resulting errors had been made in the BCS. Accordingly, model performance was good in deep waters of the BCS and medium performance was observed in the region of the Scheldt estuary. In interpreting these results, it should be noted that shallow coastal domains that interact with terrestrial water systems, such as the BCS are often prone to seasonal fluctuation of flows (Pawlowicz et al., 2002). Hence, considering fixed harmonic constants to express flow dynamicity in coastal regions might not be a right choice to do so. It may also be the case that degree of mesh refinement, detailing works and the way downstream Rivers of the Scheldt estuary were expressed in model's grid might not have been adequate. Also, considering the analysis made in section 3.3 (Figure 9), the errors encountered here might be associated with lower performance of the reference data set in the Scheldt area. Figure 5A, 5B above shows direct comparison of model and reference data sets M 2 , S 2 and M 4 constituents' harmonic constants. While, the first two semi-diurnal constituents dominate the overall tidal

Figure 6. Comparison of BCS region's tidal constituent information performance to other location of model domain. A) Analyzed locations B) Bar plot for performance of

locations.

elevations in the Belgium continental shelf area, M 4

is the dominant shallow water constituent of the

region. Accordingly, figure 5A shows that model

expresses constituent amplitudes very well and to a

similar extent, figure 5B shows that constituents phase

residuals were very small.

The next analysis was comparing models perfor

mance in different locations of the grid. Considering

the detailed mesh refinements and accurate coastal

boundary descriptions of the BCS region in model,

a reasonably better performance is expected for the

BCS region. Six locations shown in Figure 6A were

put into the analysis of which three are in deep waters

and the rest near coastal boundaries.

The bar plots provided in Figure 6B shows the

analysis results. Accordingly, deep-water locations of

the Celtic Sea, Mid-North Sea, and Irish Sea had

low Average Relative Harmonic Cumulative Errors as

compared to all coastal locations. This is true, as the

number of environmental stresses considered in com

puting the flow dynamics in the locations are much

smaller when compared to coastal locations. With Figure 7. Time series comparison between model and measurement station tide levels (Station: Bol Van Heist). Figure 8. Performance of model for high tide levels (Station: Bol Van Heist). respect to performances of costal locations in model, BCS has the highest performance. 3.2 Time series analysis of tide levels Tidal level measurements obtained from observation stations near the Belgium coastlines were compared with model results of respective location. After detailed analysis, it was concluded that Phase lags observed in model's time series were insignificant and a high degree of correlation exists between the measured and modeled tide levels. Similar results were achieved for all the five-measurement station considered in the study. Time series plot for station Bol Van Heist is provided on Figure 7 and on Figure 8 tidal peaks of Figure 7 were chosen and analyzed. Observing Figure 8 above, a good model would have a zero bias. This happens when the Bisector and Mean deviation lines fall on the same axis (when the mean deviation is zero). In terms of the upper and lower standard deviation lines, a good model would have these two lines fall on the mean deviation axis, indicating lower scattering of values along the bisector line

Figure 9. Comparison of model and derived measurement

stations harmonic constants. (Station: Bol Van Heist).

(Willems, 2009). Generally, the model performs well

but it slightly underestimates high tide levels.

3.3 Comparison of harmonic constants

This section highlights results obtained by direct

comparison of tidal constituents derived from the

calibrated tidal model, the reference data set and a

measurement station at Bol Van Heist.

The bar plots presented in the coming figures had

shown that the calibrated TELEMAC-2D model per

forms better than the European shelf data set of OTIS

in the BCS region. Similar results were achieved for

the other measurement stations too.

4 CONCLUSION

In the framework of this research, TELEMAC-2D (v7p0r1) was setup covering the entire northwestern European continental shelf with a special focus on cali brating a tidal model that will serve as a basis for future storm surge studies in the Belgium continental shelf area.

Accordingly, boundary and initial conditions were

Nicholls, Robert J., and Anny Cazenave. 2010. “Sea-Level Rise and Its Impact on Coastal Zones.” science 328(5985): 1517–20.

Ntegeka, Victor, Patrick Willems, and Jaak Monbaliu. 2012. “On the Relative Importance of the Climate Change Factors along the River Scheldt Considering Climate Sce narios for Upstream Inland and Downstream Coastal (mean Sea Level and Surge) Boundary Conditions.”, Geophysical research abstract vol. 14: EGU2012-12156.

Parker, B. B. (2007). “Tidal Analysis and Prediction”, Sil ver Spring, Maryland: National Oceanic and Atmospheric Administration.

Pawlowicz, R, B Beardsley, and S Lentz. 2002. “Classical Tidal Harmonic Analysis Including Error Estimates in MATLAB Using T_TIDE.” Computers & Geosciences (28), 929–937.

Van Lancker,V., Baeye, M., Du Four, I., Janssens, R., Degraer,

S., Fettweis, M., . . . Kom., (2012)., “Quantification of erosion/sedimentation patterns to trace the natural versus anthropogenic sediment dynamics", "QUEST4D": final

report, Belgian Science Policy Office, Brussel,103 pp. Verfaillie, E, V Van Lancker, and M Van Meirvenne. 2006. "Multivariate Geostatistics for the Predictive Modelling of the Surficial Sand Distribution in Shelf Seas." Continental Shelf Research 26(19): 2454–2468. Walters, Roy A. 2005. "Coastal Ocean Models: Two Useful Finite Element Methods." Continental Shelf Research 25(7–8): 775–93. Willems, P. 2009. "A Time Series Tool to Support the MultiCriteria Performance Evaluation of Rainfall-Runoff Models." Environmental Modelling & Software 24(3): 311–321. Woth, K, RWeisse, and HVon Storch. 2006. "Climate Change and North Sea Storm Surge Extremes:An Ensemble Study of Storm Surge Extremes Expected in a Changed Climate Projected by Four Differernt Regional Climate Models.", Ocean dynamics (56): 3–15. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Online coupling of SWAN and SWASH for nearshore applications

M. Ventroni & A. Balzano

Department of Civil, Environmental and Architecture Engineering, University of Cagliari, Cagliari, Italy

M. Zijlema

Environmental Fluid Mechanics Section, Faculty of Civil Engineering and Geosciences,

Delft University of Technology, The Netherlands

ABSTRACT: An online, one-way coupling procedure between the SWAN phase-averaged, spectral wave model

and the SWASH time domain, multi-layered non-hydrostatic model has been developed, with the aim of modeling,

seamlessly and efficiently, the wave evolution from generation to runup and land inundation. Both models can be

run either on Cartesian or curvilinear structured grids with different spatial resolution, with the SWAN domain

being typically larger than the SWASH domain. The coupling is obtained forcing the open offshore and lateral

boundaries of the SWASH domain by directional random wave fields generated through the embedded wave maker algorithm included in the SWASH code, based on the action density spectra computed by SWAN. This assures the continuity of the information, while preventing spurious reflection of outgoing waves. The coupling has been evaluated through comparison with laboratory data, with special attention paid to the location of the inner boundary between the two models.

1 INTRODUCTION

A sea state, defined by a characteristic energy spectrum, contains motions occurring at different time scales, which may range from second/minutes to days/months. A single modeling approach cannot describe accurately the entire spectrum; however, a specific combination and coupling between a phase averaged and phase-resolving model might be implemented to solve either shorter (wind waves and swell) and some longer (seiches, tsunamis, storm surges) waves at the same time. In such case, a wave-by-wave model might be used in the nearshore (say to a scale of 10 wavelengths in the cross-shore direction), while a spectral model could be extended to an oceanic scale (1000 × 1000 wavelengths). As a matter of fact, the usual dichotomy between dispersion and non-linearity may be overcome, because a coupled model would be able to compute wave transformation processes, including dispersion, as well as steepening, breaking, run-up and land inundation.

In this study, the phase-averaged SWAN model and the time-domain SWASH model have been coupled. SWAN (Simulating WAves Nearshore, Booij et al. 1999) is a third generation wave spectral model, which accounts for both offshore and nearshore processes, such as wave generation, threeand four-wave non linear interactions, bottom friction, whitecapping and depth-induced breaking. SWASH (Simulating WAves till SHore, Zijlema et al. 2011) is a multi-layered, non

hydrostatic flow model, which is a reasonable and efficient approximation of the Reynolds-averagedNavier-Stokes equations, ensuring a good balance between nonlinearity and frequency dispersion. In fact, its numerical dispersion increases with the number of layers. Introducing the wave number k , the angular frequency σ and the water depth d, it may be shown that with K vertical equidistant layers the frequency dispersion is represented with a $[2K-2,2K]$ Padè expansion in $\mu = kd$ of the expression for the phase velocity σ/k (Zijlema & Stelling 2008). This implies that one, two and three layers are sufficient to compute linear dispersive waves, respectively, up to $\mu = 0.5$, 7.7 and 16.4 with a relative error of at most 1% in phase velocity. However, the computational effort required to simulate intermediate to large areas at the time scales of interests is still prohibitive within the engineering context (Smit et al. 2013). On the other hand, this computational restriction can be alleviated at the expense of reducing the overall accuracy, with a phase-averaged approach. Consequently, a reasonable solution would be to carry out practical applications using the non-hydrostatic model in synergy with the spectral model. A preliminary validation to the application of this coupled model represents the intent of this study. To this aim, results of four laboratory tests on wave run-up over a gently sloping beach were used (Mase 1989). The outline of this paper is as follows. First, the main characteristics of SWAN and SWASH are presented in section 2. The implementation and mechanism of the

coupling are reported in section 3, while section 4 deals

with the application of the coupled model. Finally, in

section 5 conclusions are reported.

2 MODEL COMPONENTS

2.1 SWAN

The spectral phase-averaged model used in this study

is the third-generation open source wave model SWAN

version 41.01.The model describes the evolution of the

action density N , defined as the ratio of energy spectral

density to the relative angular frequency, numerically

solving with an implicit scheme the spectral action

balance equation, which reads (Booij et al. 1999):

The left-hand side is the kinematic part of this

equation. The second term denotes the propagation of

wave action in the two-dimensional physical space,

with c g the group velocity and U the ambient current.

The other two terms represent the energy shift in the

spectral space (σ,υ), due to depth-induced and current

induced refraction, with υ indicating the propagation

direction. The right-hand side contains the dynamic

part of the equation, where S tot is a source/sink term

representing all physical processes that generate, dis

sipate and redistribute wave energy. Six processes

contribute to S tot :

These terms denote, respectively, energy input by wind, nonlinear transfer of wave energy through three wave and four-wave interactions, wave decay due to whitecapping, bottom friction and depth-induced wave breaking. Extensive details on the formulation of these processes can be found in Booij, et al. (1999) and Holthuijsen (2007). Recently, new formulations of depth-induced wave breaking have been added, Salmon et al. (2015).

In order to obtain a unique solution of Equation 1, boundary conditions should be provided. The incom ing wave components at the seaward boundaries are specified by a directional spectrum. The land bound aries are fully absorbing for wave energy leaving the geographical domain. The lower and upper boundaries in the frequency space, indicated by σ min and σ max , respectively, are fully absorbing. However, a f −4 diag nostic tail is added above the upper cut-off frequency, which is used to compute nonlinear wave-wave inter actions and for computing integral wave parameters. No boundary conditions are needed in the directional

space. 2.2 SWASH 2.2.1 Governing equations The SWASH model is a multi-layered, non-hydrostatic flow model, which solves numerically the Reynoldsaveraged Navier-Stokes equations for an incompressible fluid with a constant density and a free surface.The 3.14 open-source version is used in this study. We limit the discussion to the one-dimensional framework, with straightforward extension to the other horizontal direction. The governing equations read: where x and z are Cartesian co-ordinates (z = 0 is located at the still water level), t is time, u(x, z, t) is the horizontal velocity, w(x, z, t) is the vertical velocity; ρ is density; p h and p nh are the hydrostatic and nonhydrostatic pressure components, respectively, and τ xx , τ xz , τ zz , τ zx are turbulent stresses. The domain is bounded vertically by the free surface at z = ζ(x,t) and the bottom at z =−d(x). The hydrostatic pressure is expressed in terms of the free surface elevation as p h = ρg(ζ-z) so that ∠ z p h =−gρ and ∠ x p h = ρg∠ x ζ, with ∠ i =∠/∠ i | i=x,z , g being the gravitational acceleration. Kinematic boundary conditions are prescribed at the free surface and bottom, given by: These boundary conditions ensure that particles laying on the free surface or on the fixed bottom do not leave those surfaces. By integrating Equation 3 over the water depth h = ζ + d and using the kinematic conditions, Equations 6–7, the free surface condition is obtained: At the free surface the dynamic boundary condition is constant atmospheric pressure (i.e. zero relative pressure, p h = p nh = 0) and no shear stresses. At the bottom boundary a bottom stress term is added to the horizontal momentum Equation 4, based on a quadratic friction law where U is the depth averaged current and c f is a dimensionless friction coefficient.

Appropriate boundary conditions need to be imposed at the open and closed boundaries of the com putational domain.At the offshore boundary incoming irregular waves are introduced by specifying a verti cal profile of horizontal velocities. These velocities are either obtained from time-series for each point on the boundary, or by imposing a wave spectrum. Addi tionally, to simulate entering waves and to allow low frequency energy to leave the domain, a weakly reflec tive condition acting on outgoing waves is adopted (Blayo & Debreu 2005), assuming that incoming and outgoing waves are perpendicular to the boundary. At closed boundaries zero normal velocity and free-slip conditions are imposed.

2.2.2 Numerical implementation

The physical domain is discretized on a structured, staggered grid with constant grid size in the horizontal planes, whereas a boundary-fitted grid is employed in the vertical. Actually, the total water depth h is split up defining a fixed number K of terrain-following layers with a spatially varying thickness $\Delta z = h/K$.

The numerical implementation is based on an explicit, second order accurate (in space and time) finite difference method for horizontal momentum, with a pressure correction technique employed to com pute the non-hydrostatic pressure, ensuring both local and global mass and momentum conservation at the numerical level. A more detailed description of the numerical implementation can be found in Zijlema

In the surf zone, SWASH intrinsically accounts for the energy dissipation of a breaking wave, at a rate analogous to that of a bore, and is able to reproduce accurately the actual location of incipi ent wave breaking. This feature is rooted into the shock-capturing property of the model, due to the implemented momentum-conservative scheme. How ever, high vertical resolutions is required to achieve such accurate results, whereas at low resolution wave breaking is delayed (Smit et al. 2013). To capture wave breaking with only a few vertical layers, Smit et al. (2013) proposed an approach with which the non-hydrostatic pressure is neglected in the vicinity of a breaking wave. This (locally) reduces the govern ing equations to the nonlinear shallow water (NLSW) equations and ensures that a wave develops a vertical face. This approach, defined as the hydrostatic front approximation, is initiated once the rate of change of the free surface exceeds a threshold fraction α of the long wave celerity ($\partial_t \zeta > \alpha (gh)^{1/2}$). Once initiated, α is reduced to a lower value β in neighbouring points to mimic breaker persistence. In this study, the default values, found after calibration by Smit et al. (2013) for the maximum steepness parameter (α= 0.6) and the persistence parameter (β= 0.3), were used.

Finally, the reliable and simple wet-dry approach presented in Stelling & Duinmeijer (2003) is imple mented.This method tracks the motion of the shoreline by ensuring non-negative water depths and using the upwind water depths in the momentum flux approxi mations. 3 COUPLING PROCEDURE A code producing a single executable was implemented, where SWAN was considered as the master model, whereas SWASH was defined as a set of subroutines packed into a library, that can be called inside the SWAN main time loop. The two models are applied to different structured domains, either Cartesian or curvilinear, with SWAN acting on a larger domain, which is extended up to offshore, while the SWASH domain is located in the nearshore and, possibly, includes the emerging beach terrain. The models run sequentially in time. The coupling is accomplished prescribing the variance density spectrum computed by SWAN to the SWASH open boundaries, either offshore and/or lateral, representing an inner boundary for the broader domain of the coupled model. The SWAN spectrum is sampled using the wave-maker algorithm included in the SWASH model (Zijlema et al. 2011, Smit et al 2013), which is based on a single-summation method (Miles 1989) and produces a quasi-homogeneous wave variance (Miles & Funke 1989). The coupling is one-way in that the action balance equation model does not take into account the offshore directed, reflected waves computed by the phase resolving model. Furthermore, the wave-induced setup computed by SWAN can be passed to SWASH. 4 APPLICATION Given the heavy computational requirements of SWASH, a crucial point in setting up the coupled model is represented by the optimal location of the inner boundary, so as to reduce as much as possible the SWASH domain and related computational burden without losing the outcome’s accuracy and reliability. Namely, the tradeoff is between having the inner boundary far enough from the shore to represent the wave nonlinearity wherever significant, but not as much as to include areas of large wave dispersion, requiring too many layers to be represented. To investigate this point, four test cases are selected from Mase’s (1989) laboratory dataset (Tab. 1). The results may be compared with McCabe et al. (2010, 2011) findings, which used the same tests to investigate the best coupling location within a coupled SWAN-NLSW model. To this aim, SWAN, SWASH and SWAN + SWASH model runs were carried out. 4.1 Test description Mase (1989) studied random wave runup for a range of wave conditions and four test slopes of a model beach. The experimental setup can be seen in Figure 1 of Mase & Iwagaki (1984). Four of these tests with bed slope m = 1:20 and water depth at the wavemaker d wm = 0.45 m will be considered here, with wave conditions at the wavemaker and breaker types given in Table 1, where H wm is the significant wave height,

Figure 1. SWAN results. Computed wave height (first panel) and setup (second panel), normalized with respect to the

wavemaker depth, fraction of breaking waves (third panel) and coupling points along the bottom profile (last panel), for each

test case.

Table 1. Parameters of incident waves at the wavemaker for the Mase (1989) tests.

Case	H_{wm} (cm)	T_p (s)	L_{wm} (cm)	ξ_{wm} (-)	Breaker type
TEST A	4.95	2.50	500	0.702	Plunging
TEST B	6.18	2.00	388	0.502	Spilling
TEST C	7.37	1.67	312	0.384	Spilling
TEST D	9.14	1.25	212	0.258	Spilling

T_p the peak period, L_{wm} a representative wavelength calculated with linear theory for a monochromatic wave of period T_p, and $\phi_{wm} = m/(H_{wm}/L_0)^{1/2}$ is the surf similarity parameter or Iribarren number,in which $L_0 = gT_p^2/2\pi$ is the offshore wavelength.

Comparison between the measured and calculated runup statistics will be made. The numerical model is set-up along a one-dimensional flume of length L_x = 25 m, with horizontal resolution Δx.

SWASH was run with time step Δt for a duration D = 1200 s, with spin-up time D_{su} = 300 s. At the

Table 2. Numerical setup used in SWASH runs.

Case	Δx (m)	Δt (s)
TEST A	0.020	0.002
TEST B	0.020	0.002
TEST C	0.015	0.001
TEST D	0.010	0.001

wave maker boundary a Pierson-Moscowitz spectrum is imposed. For case specific model parameters we refer to Table 2. Each test case was run in three different configurations: (i) one vertical layer, (ii) the same as (i) including bound waves (Rijnsdorp et al. 2014) in the seaward boundary condition, (iii) two vertical layers. Turbulence and bottom friction are neglected. Following McCabe et al. (2011), the SWAN model was run without nonlinear interactions (quadruplets and triads) and bottom friction, but activating whitecapping and depth-limited breaking (Battjes & Janssen, 1978). The frequency space was subdivided

Figure 2. Errors in runup statistics, compared with Mase's (1989) results, with SWASH model used throughout the domain,

with one, or three layers, and with one layer plus bound waves at the wavemaker, as a function of the surf similarity

parameter, ξ_{wm}.

in 200 parts, with lower and upper boundaries chosen

equal to $0.5f_p$ and $3f_p$, respectively, $f_p = 1/T_p$ being the

peak frequency. Finally, the long crestedness is sim

ulated with a $\cos^{800}(\theta)$ directional distribution, cor

responding to a standard deviation of the directional

distribution (Kuik et al. 1988), $\sigma_\theta = 2^\circ$.

The coupled SWAN + SWASH model was run with

one layer only, with the same D and D_{su} used for

SWASH alone. For each test, six to seven locations of

the coupling points were tested, with values of H_{m0}/d

in the range 0.1–0.6, d being the undisturbed depth, i.e.

not accounting for wave setup. For each of the resulting

combinations, five runs were carried out with different

random seeds used in the generation of random input

phases of the wave components at the inner boundary,

to which the results may be sensitive.

Incoming waves are imposed at the SWASH open

boundary by prescribing layer-integrated horizontal

velocities computed from the hyperbolic cosine verti

cal profile given by linear theory. The same approach was used in the coupled model, where the coupling

is obtained at the inner boundary with a sending

receiving process, such that: (i) the spectrum computed

by SWAN in conjunction with a weakly reflective

boundary condition are imposed; (ii) waves are syn

thesized by using a single summation method, with a

single random wave angle assigned to each frequency

(Miles 1989); (iii) the horizontal velocity, computed

using the linear theory and taken in direction normal

to the SWASH boundary, is prescribed.

Furthermore, in the coupled model, wave-induced

setup calculated by SWAN, integrating the 1D equa

tion for the average level forced by the radiation

stresses, is passed to the grid points of the phase

resolving model. Although, as a spectral model, SWAN does not represent physical processes at scales less than a wave length, SWAN uses the same grid as SWASH in the nearshore to share information between the models efficiently in the coupled scheme. Albeit the simulation is stationary, with both models being thus run once, this can be accomplished because the models run sequentially, with SWAN running first from offshore to inshore, while SWASH is run from the coupling point shoreward, but knowing average depths and levels computed by the phase-averaged model at the time of coupling. The setup computation made by SWAN can thus be seen as a sort of predictor of the setup computed by SWASH. Lastly, no ad-hoc boundary condition is required at the shoreline, because swash and related moving boundary are explicitly represented through accurate and efficient flooding and drying handling. A threshold value of 0.1 mm was set in each run to represent the minimum inundation depth. Of course, it was also checked that the moving boundary never reached the end of the computational domain. 4.2 Results and discussion In Figure 1 the significant wave height and the waveinduced setup are shown, normalized with respect to the depth at the wave-maker, as well as the fraction of breaking waves as computed by SWAN. This latter is helpful in choosing the coupling points, considered within the coupled model, which are shown in the last subplot. Figure 2 shows the relative errors in runup statistics $R_{2\%}$, $R_{1/3}$ and $R_{1/10}$ with respect to Mase's experimental data, as a function of the surf similarity parameter ξ_{wm} calculated with SWASH throughout

Figure 3. Errors in runup statistics for the coupled SWAN+SWASH model, compared with Mase's (1989) laboratory results,

as a function of H_{m0}/d at coupling location. Errors are mean values of 5 randomly phased wave trains prescribed as boundary

conditions of SWASH, generated from the same action density spectrum computed by SWAN.

the computational domain in three different configura

tions: with a single layer, a single layer accounting for

bound waves at the wavemaker, and with three layers.

Note that testsA to D are in the order of decreasing ξ_{wm}.

Although the effect of increasing the number of

layers is apparent, three layers still seem to be too

little to ensure adequate representation of wave dis

persion with the shortest and steepest waves (test D,

$d_{wm}/L_{wm} = 0.21$, $\tanh(2\pi d_{wm}/L_{wm}) = 0.87$), resulting

in some 20% error on $R_{2\%}$, with smaller errors obtained

for $R_{1/10}$ and $R_{1/3}$.

Figure 3 shows the relative errors with respect to

the experimental data, calculated with the coupled

SWAN + SWASH model as a function of the non

linearity parameter at the coupling point, H_{m0}/d, to investigate how the location of the coupling point influences the accuracy of results with respect to the complete phase-resolving run. The errors of the coupled model were computed as average values over five runs with randomly phased wave trains prescribed as boundary condition of SWASH, generated from the same action density spectrum computed by SWAN.

Each run was performed using the SWASH model with a single layer, in order to test the accuracy of the coupled model with the less computationally expensive configuration.

From comparison between Figures 2–3, it can be seen that results in the coupled model can be improved varying the location of the coupling point. For example, the model performance inTest D is enhanced either using more layers, but increasing the computational time, or choosing a coupling point closer to the shore, with a significantly lower computational effort. On the other hand, it is apparent that the choice of the coupling point is not unique for the wave conditions analyzed, nor the error–H_{m0}/dcurves are monotone. The optimal coupling point depends also on the accuracy of the SWAN spectral modeling. However, it seems that a common value of$H_{m0}/d \approx 0.55$ might be chosen for test cases B, C, D, while a lower value of $H_{m0}/d \approx 0.3$ appears to be the best in Test A, differing from the other tests as to breaker type. These values are lower than McCabe et al. (2011) results, where a value of $H_{m0}/d \approx 0.65$ was found to represent the optimal switching point in their SWAN+NLSW model. This is probably due to SWASH being still able to represent some degree of wave dispersion even with one layer compared to the fully non-dispersive NLSW formulation. It is also to be noted that taking the coupling point too much shorewards resulted in large errors or instability. On balance, simulations are shown to be reasonably effective with the single SWASH model used throughout for sufficiently large peak periods, and in a broader peak period range with the coupled model, for the wave conditions studied. 5 CONCLUSIONS In the present study, an online, one-way coupling between the SWAN and SWASH models has been introduced. Results show a reasonably good agreement with the set of runup statistics of random wave

experimental data, either using the SWASH or the

coupled model.

It is shown that the SWASH domain can be reduced

shifting the coupling point shorewards, consequently

reducing the computational effort, while still obtaining

reliable results.

The choice of the optimal coupling point is found

to be dependent on the wave conditions analyzed, with

a common location expressed by the value $H_{m0}/d \approx$

0.3–0.55.

Running a wider variety of test cases with differ

ent model configurations is required to fully validate

and improve the proposed approach. Namely, the sim

ulation might be done using the SWAN model with

a different depth-induced wave breaking (see Salmon

et al. 2015 for a range of possibilities) and/or includ

ing the transfer of energy due to triad nonlinear

interactions.

The proposed approach will be further extended.

Specifically, a parallel version will be implemented and a tight coupling, which will assure stronger feed

back between models, will be realized in the near

future.

Furthermore, the efficiency of the coupled model

will be improved.

ACKNOWLEDGEMENTS

The first author gratefully acknowledges Sardinia

Regional Government for the financial support of

his PhD scholarship (POR Sardegna FSE Operational

Programme of the Autonomous Region of Sardinia,

European Social Found 2007-2013 - Axis IV Human

Resources, Objective I.3, Line of Activity I.3.1.).

The research was also funded by the Regional

Agency for the Sardinian Hydrologic District

(ARDIS) - contract Prot. 11554 Rep 4/2012: Regional

Mapping of Coastal Flooding Hazard.

Battjes, J. A. & Janssen, J. P.F.M. 1978. Energy loss and set

up due to breaking of random waves. Proc. 23rd Intern.

Conf. Coast. Engrg.: 42-50. New York: ASCE.

Blayo, E. & Debreu, L. 2005. Revisiting open boundary con

ditions from the point of view of characteristic variables.

Ocean Modelling 9: 231-252. Booij, N., Ris, R.C. & Holthuijsen, L.H. 1999. A thirdgeneration wave model for coastal regions. 1. model description and validation. Journal of Geophysical Research 104(C4): 7649-7666. Holthuijsen, L.H. 2007. Waves in Oceanic and Coastal Waters. Cambridge: Cambridge University Press. Kuik. A.J., Van Vledder, G.P. & Holthuijsen, L.H. 1988. A method for the routine analysis of pitch-and-roll buoy wave data. Journal of Physical Oceanography 18: 1020– 1034. Mase, H. & Iwagaki, Y. 1984. Run-up of random waves on gentle slopes. Proc. 19 th Coastal Engrg. Conf.: 593–609. Houston: ASCE. Mase, H. 1989. Random wave runup height on gentle slope. Journal of Waterway, Port, Coastal and Ocean Engineering 115(5): 649–661. McCabe, M. & Stansby, P.K. 2010. An investigation on the coupling of spectral energy modeling and Boussinesq type modeling in the nearshore. Proc. 1 st European IAHR Congr.. Edinburgh: IAHR. McCabe, M., Stansby, P.K. & Apsley, D.D. 2011. Coupled wave action and shallow-water modeling for random wave runup on a slope. Journal of Hydraulic Research 49(4): 515–522. Miles, M.D. 1989. A note on directional random wave synthesis by the single-summation method. Proc. 23rd IAHR Congress.: C-243-C-250. Ottawa: The National Research Council Canada. Miles, M.D. & Funke, E.R. 1989. A comparison of methods for synthesis of directional seas. Journal of Offshore Mechanics and Arctic Engineering 1110 (1): 43–48. Rijnsdorp, D.P., Smit, P.B. & Zijlema, M. 2014. Nonhydrostatic modelling of infragravity waves under laboratory conditions. Coastal Engineering 85: 30–42. Salmon, J.E., Holthuijsen, L.H., Zijlema, M., Van Vledder, G.P. & Pietrzak, J.D. 2015. Scaling depth-induced wavebreaking in two-dimensional spectral wave models. Ocean Modelling 87: 30–47. Smit, P., Zijlema, M. & Stelling, G.S. 2013. Depth-induced wave breaking in a non-hydrostatic, near-shore wave model. Coastal Engineering 76: 1–16. Stelling, G.S. & Duinmeijer, S.P.A. 2003.A staggered conservative scheme for every Froude number in rapidly varied shallow water flows. International Journal for Numerical Methods in Fluids 430(12): 1329–1354. Zijlema, M. & Stelling, G.S. 2008. Efficient computation of surf zone waves using the nonlinear shallow water equations with non-hydrostatic pressure. Coastal Engineering 55: 780–790. Zijlema, M., Stelling, G.S. & Smit, P. 2011. SWASH: an operational public domain code for simulating wave fields and rapidly varied flows in coastal waters. Coastal Engineering 58: 992–1012. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Spectral parameters modification in front of a seawall with wave return

Th. Giantsi, G. Papacharalampous, N. Martzikos & C.I. Moutzouris

Laboratory of Harbour Works, School of Civil Engineering, National Technical University of Athens, Greece

ABSTRACT: Measurements of wave characteristics were performed on a physical model of a seawall with wave return. The observations reveal modifications in the shape of the spectra, in the spectral broadness ε, the narrowness ν and the peakedness Q_p parameters. These modifications depend on the Ursell number (U_r) and on geometrical parameters such as the water depth (h) and the distance from the front of the structure (D_L). The parameters ε and ν are proportional to wave height, wave length, D_L and h. The parameter Q_p depends on the wave length, the distance from the structure and the water depth. The broadness parameter ε shows the best correlations among the considered parameters. In some cases, a double peaked spectrum was observed near the seawall, with a displacement of the peak period towards a lower frequency.

1 INTRODUCTION

Seawalls with wave return are vertical structures constructed to protect the inland from the wave action and most often from the wave overtopping. Seawalls with wave return affect also incident waves by reflecting them, causing navigation problems. Previous studies in this topic mainly focused on wave overtopping (Owen & Steele 1993, Besley 1999, Pearson et al. 2005, Giantsi et al. 2012). The wave reflection on such structures has also been investigated by Giantsi et al. (2014), correlating the wave reflection and geometrical characteristics of the structures.

Time domain analysis of wave measurements was used by Ohle et al. (2005), Boccotti et al. (2013), and Boccotti (2013), to estimate the wave transformation perpendicular to the shoreline under the influence of vertical structures. Wave skewness and asymmetry were investigated by Peng et al. (2009) and Zou & Peng (2011) in cross-shore evolution, under the presence of low-crested breakwaters, or over a natural inter-tidal sandbar by Brinkkemper (2013). Rocha et al. (2013) investigated the wave nonlinearity using field data.

In the present study, wave characteristics were measured in front of a physical model of a composite seawall with wave return.The measurements were performed in the direction perpendicular to the structure and at specific distances from the structure. The spectral analysis revealed modifications in the shape of the measured spectra. The focus of the present study was the influence of the structure on the spectrum of the nearby wave field.

2 METHODOLOGY

2.1 Physical model

To investigate the performance of a composite seawall with wave return, a 3 m-long physical model was constructed in a wave basin at the Laboratory of Harbour Works, National Technical University of Athens. The dimensions of the wave basin are 24 m × 26 m × 1.1 m. For the needs of the present re-search, only a part of the basin, with dimensions 7.5 m × 26 m × 1.1 m, was used with one wave generator.The physical model of the seawall was constructed on a horizontal bottom, reproducing the foundation of the structure only till a certain depth. Considering a geometrical scale of 1:30, the depth ranged between 15 and 16 meters. The waves were measured in front of the seawall, at four locations, to calculate the wave reflection for different wave conditions. A 5th wave gauge was located near the wave generator to control the incident wave. Two different sections (A and B) of the composite seawall were tested under a combination of water levels and wave conditions. Figure 1 shows a typical cross section of the experimental layout in the wave basin, with the physical model and the wave generator. Geometrical dimensions of the two tested sections (A and B) are given in Table 1. The dimensions included in Table 1 are the water depth in the basin (h), the freeboard (R_c), the radius of the wave return curvature (R) and the upper vertical part of the wave return (B_c). The seawall was founded at a riprap with a protecting toe. Detail of the typical cross section B is given in Figure 2. For the reproduction of the waves, a paddle wave generator was used, producing JONSHAP type spectra with a peak enhancement factor of 3.3. Resistive type wave gauges collected wave data under a 50 Hz sampling rate throughout the entire 600 sec duration of each test. Five wave gauges were used, 4 of them (Pr1 to Pr4) were placed in front and perpendicular to the composite structure and one (Pr5) in the area of wave generation, at a distance of 2 m from the wave generator paddle. Cross section of the physical model with the locations of wave gauges is presented in

Figure 1. Typical cross section of the experimental layout.

Table 1. Dimensions of sections tested A and B.

TEST SERIES	CREST * LEVEL (m)	h (m)	R_c (m)	R (m)	B_c (m)
A.1	0.633	0.500	0.133	1.1	0.8
A.2	0.633	0.523	0.100	1.1	0.8
A.3	0.633	0.513	0.121	1.1	0.8
B.1	0.653	0.500	0.153	1.2	0.65
B.2	0.653	0.513	0.14	1.2	0.65
B.3	0.653	0.520	0.133	1.2	0.65

B.4 0.653 0.533 0.120 1.2 0.65

B.5 0.653 0.543 0.100 1.2 0.65

* The crest level is measured from the bottom of the basin

Figure 2. Detail of the cross section B.

Figure 3. The distances between the wave gauges and

the structure are detailed in Table 2.

For each data set under the same water level, 60

tests were conducted. Six wave peak periods were

tested, from T_p = 0.786 s to T_p = 1.406 s. For each

wave period, five different wave heights were tested.

Each wave condition was tested twice.

Dimensional and dimensionless spectral parame

ters were calculated using the wave measurements

spectral analysis. The dimensionless parameters were

correlated to the ratios of dimensional and geomet

rical parameters as water depth (h), distance from the

structure (D_L), freeboard (R_c) and shape of the seawall. Figure 3. Layout of the model. Table 2. Distances of the wave gauges from the structure. Distance between Distance from Point (m) the structure (m) Structure 0.00 0.00 Pr4 1.25 1.25 Pr3 0.35 1.60 Pr2 0.45 2.05 Pr1 0.20 2.25 2.2 Spectral analysis Performing spectral analysis on time series of sea surface elevation provides the moments m (zero to nth order), which describe the spectrum. Using the moments, dimensional and dimensionless parameters were calculated to provide further information. Common dimensional spectral parameters are the significant wave height H_s, the mean period T_m, and the wavelength L_m, with: To better describe the spectral shape, three main dimensionless shape parameters were used: the broadness parameter ε, the narrowness parameter ν and the peakedness parameter Q_p, with:

Figure 4. a,b Spectra in front of section A.

Figure 5. a, b Spectra in front of section B.

These spectral shape parameters are correlated to the Ursell number (U_r), describing the non-linearity of the waves.

3 ANALYSIS OF THE RESULTS

3.1 Wave spectra

Wave measurements were analyzed using "HR Wave Data – Data Acquisition and analysis software program'' (Beresford et al., 2006), with Fast Fourier Transformation method. Wave spectral and statistical parameters were obtained.

Typical plots of the produced wave spectra from the 5 locations of wave measurements, are presented in Figures 4 a, b, for section A and Figures 5a, b, for section B.

Modifications from the initial shape of the incident wave (from probe 5) were observed.

a) The peak period T_p measured by the wave gauges, at the nearest to the structure location, was moved to lower frequencies (higher periods) or the spectrum transformed to a double peak. b) In some locations, the maximum energy density exceeds its value in the area of wave generation near the paddle by about 20–30%. c) In some cases, a modification of the frequencies was observed, with a reduction in the higher frequencies and a displacement in the lower part of the spectrum, or the opposite. d) The most severe modifications were observed in steep waves and at the locations close to the structure. e) Spectra from locations Pr1 and Pr2 had similar behavior. They were more uniform. 3.2 Shape spectral parameters The spectral shape parameters, as the broadness parameter ε, the narrowness parameter ν and the peakedness parameter Q_p. were calculated using the moments and the outcomes of the spectral analysis. To investigate their behavior, the dimensionless Ursell number was used first. The distribution of these

Figure 6. The parameter ε versus Ursell number for: a) section A, b) section B.

Figure 7. The parameter ν versus Ursell number for: a) section A, b) section B.

parameters along an experimental channel was inves

tigated by Zhang H., (2011).

Figures 6 to 8 present the parameters ε, ν and Q_p

plotted versus the Ursell number, for the Sections A

and B.

In order to investigate the relationship between

the above parameters and the distance from the

structure, the dimensionless parameter $d*$ (inversely

proportional to the distance D_L) was introduced, as

follows:

The shape parameters ε and ν are plotted versus d* in

Figures 9 and 10.

Due to non-satisfactory correlation results for the

parameter Q_p, it was decided to use another dimension

less ratio, independent of the wave height, and defined as the Ursell number multiplied by the ratio D_L/H_s. (Figure 11) As it is observed from Figure 6, the broadness parameter ε is well correlated to Ursell, for each data set separately, at every location of measurement. The range of parameter ε values is between 0.33 and 0.85. The behaviour of both sections A and B is similar. Longitudinal trend lines were implemented with very high correlation factors R2, ranging between 0.74 and 0.92 for all data sets. Better correlation factors were obtained at data sets from Section

A, for all locations. The trend line from the data set at the location closest to the structure (Pr4) shows the steepest slope, when the Ursell number is greater than 1.0, corresponding also to the maximum values of ε. Data from the control wave gauge (Pr5), in the vicinity of the wave generator, give also a steep trend line. Data with an Ursell number greater than 1.0 are more dispersed than the others for all measurements. The Ursell number is

Figure 8. The parameter Qp versus Ursell number for: a) section A, b) section B.

Figure 9. The parameter ε versus d * for: a) section A, b) section B.

Figure 10. The parameter ν versus d * for: a) section A b) section B.

Figure 11. The parameter Qp versus Ur x(DL/Hs) for: a) section A b) section B.

Figure 12. The parameter ε for both sections, versus a) theUrsell number, b) d * .

Figure 13. The parameter ν, for both sections, versus a) theUrsell number, b) d * .

Figure 14. The parameter Q p for both sections, versus a) the Ursell number, b) U r x(D L /H s).

below 3 for data sets from the location Pr4 and less than 4 for the location Pr5. Because of the constant water depth, small Ursell numbers led to smaller wave characteristics.

The narrowness parameter ν is also correlated to Ur, for each measurement location separately (Figure 7). The cloud of parameter ν data points is similar to the cloud of parameter ε data points. The range of ν values is between 0.15 and 0.45. Data with Ur > 4.0 seem independent of U r .

The peakedness parameter Q_p has a different behaviour as a function of the Ursell number, than the two other shape parameters (Figure 8). The slope of the data trend line is negative for $U_r < 1.0$ and then the data seems to be independent of U_r. Data from measurement locations Pr1, Pr2 and Pr3, in the middle of the channel, show higher values of Q_p, while the remaining, in front of the structure and near the paddle, had lower values. Values of Q_p ranged from 1.85 to 5.2 for both tested sections.

The broadness parameter ε seems to be very well correlated to the parameter $d*$, even better than the Ursell number, as data from all locations had the same behaviour with a uniform cloud of points (Figure 9). For the narrowness parameter ν, data points from the closest location to the structure (Pr4) differ from the others (Figs.10a, b). The smallest waves at the longest distances give the smallest values of parameters ε and ν.

The peakedness parameter Q_p was found independent of $d*$ and for this reason was correlated to the dimensionless product as described above (Figure 11). Therefore Q_p seems to depend on the wavelength, the distance from the structure and the water depth.

3.3 Comparison of the sections

The most important difference between the geometry

of the two tested sections was the total height (water

depth plus freeboard) of the structure: section B was

higher by 20 mm than the section A (653 mm total height of section B instead of 633 mm total height of section A). Comparing the two data sets of wave measurements in front of the alternative sections of the seawall, the influence of the freeboard will be the most important factor. Data sets of parameters ε, ν and Q_p from both tested sections correlated to the above mentioned parameters (U_r , d_* , U_r x (D_L /H_s) are presented in Figures 12 to 14. All comparisons of the respective data sets from the two sections present the same trends. For section B, five series of experiments were executed, while only three series were performed for section A. The broadness parameter ε is very well correlated to the parameter d_* and to the Ursell number (Figure 12). Trend lines obtained between ε and d_* show a satisfactory correlation factor $R^2 = 0.81$ and $R^2 = 0.71$ for data sets of sectionA and B respectively.The parameter ε is influenced also by the geometrical characteristics of the sections. The taller structure (Section B) gives a little higher values of ε compared to the values of the shorter Section A. The same trends are observed at the narrowness parameter ν, (Figure 13) without an acceptable correlation factor for simple trend lines. The broadness parameter Q_p depends on the wavelength, the distance from the structure and the water depth. For all tested parameters the higher values are observed at section B and the lowest at section A. 4 CONCLUSION In the present study the main question was the influence of a seawall with wave return on the shape of the wave spectrum and on its shape parameters, in front of the structure. As regards the modifications of the spectrum shape, deformation and displacement were observed in the

energy density and frequencies. More detailed inves

tigations and analyses are still needed.

The shape parameters ε, ν and Q_p , are strongly

dependent on the wave and structure characteristics,

including the water depth and the distance from the

structure. More specifically:

• The parameters ε and ν depend on Ursell number and the parameter $d*$. The parameter $d*$ is proportional to the wave height, the wave length and inversely proportional to the distance from the structure and to the water depth.

• The parameter Q_p is proportional to the wave length, to the distance from the structure, inversely proportional to the water depth and it is independent of the wave height.

Further research is needed in order to incorporate the influence of the structure shape, including the freeboard, on the spectrum shape and on the spectral parameters also.

Beresford P.J., Spencer J.M.A and Clarke J., 2006. “HR Wave Data – Data Acquisition and analysis software program”. User manual. Report IT 493. H.R. Wallingford.

Besley, P., 1999, Overtopping of seawalls – design and assessment manual. R&DTechnical ReportW 178, Environment Agency, UK (ISBN 1 85705 069 X).

Boccotti P., Arena F. and Fiamma V.,2013, Distributions of Wave Heights in Time Domain in Stationary Sea States, Journal of Waterway, Port, Coastal, and Ocean Engineering, American Society of Civil Engineers, V. 139, pp.147-156.

Boccotti P.2013, Field Verification of Quasi-Determinism

Theory for Wind Waves Interacting with Vertical Break

water, Journal of Waterway, Port, Coastal, and Ocean

Engineering,American Society of Civil Engineers, V 139,

pp.358–364. Brinkkemper, J.,2013, Modeling the cross-shore evolution of asymmetry and skewness of surface gravity waves propagating over an intertidal sandbar, MSc Thesis, Utrecht University, The Netherlands. Giantsi, Th. E., B Papathanassiou, S. Azorakos and Moutzouris C.I.,2012, “Wave overtopping of the new runway at Thessaloniki airport”, Fourth International Conference of Physical modelling to Port and Coastal Protection, Ghent, 17–20 September 2012. GiantsiTh., PapadopoulosA. and Moutzouris C.I., 2014, “3D Experimental investigation of wave reflection on a vertical seawall with wave return”, 3rd IAHR Europe Congress, Book of Proceedings, Porto – Portugal. Ohle N., Daemrich K.-F., and Tautenhain F., 2005, Influence of Spectral Shape on wave Parameters, and design methods in time domain, Ocean Wave Measurements, and Analysis, Fifth, International Symposium Waves 2005, Madrid, Spain. Owen M.W. and SteeleA.A.J., revision 1993, Effectiveness of recurved wave return walls, Report 261, HR Wallingord. Pearson J., Bruce T., Allsop W., Kortenhaus A. and Van der Meer J.W., 2005, Effectiveness of recurved walls in reducing wave overtopping on seawalls and breakwaters, Proceedings of 29th Inter. Conf. on Coastal Engineering ICCE 2004, Lisbon, 4404–4416. Peng Z., Zou Q.P., Reeve D. and Wang B., 2009, Parameterisation and transformation of wave asymmetries over a low-crested breakwater, Coastal Engineering 56 (2009) 1123–1132. Rocha, M.V.L., Silva, P.A., Michallet, H., Abreu, T., Moura, D., Fortes, C.J., 2013. Parameterizations of wave nonlinearity from local wave parameters: a comparison with field data, Proceedings 12th International Coastal Symposium (Plymouth, England), Journal of Coastal Research, Special Issue No. 65, pp. 374–379. Zhang H., 2011, Analysis of Laboratory Generated Sea Waves, Master Thesis, Universidade Tecnica de Lisboa. Zou Q.P. and Peng Z., 2011, Evolution of wave shape over a low-crested structure, Coastal Engineering 58 (2011) 478–488. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Renewal time scales in tidal basins: Climbing the Tower of Babel

Daniele P. Viero & Andrea Defina

Department of Civil, Architectural and Environmental Engineering (ICEA)

University of Padova, Padova, Italy.

ABSTRACT: Transport time scales are key parameters to assess water renewal of estuaries and tidal basins

and they have long been used to this purpose. In a modern-day Tower of Babel, an ever-increasing confusion has

involved terminology, use and estimation of time scales, despite valuable efforts towards a sounder theoretical

framework. Within the framework of tidally flushed, semi-enclosed basins with negligible freshwater inflow,

water renewal is strongly controlled by diffusion. The inlet of a tidal basin acts alternatively as a source and as

a sink. A significant fraction of effluent water can return to the basin each flood tide. Accordingly, to correctly

assess water renewal of such basins, a region larger than the basin itself has to be considered. To account for

the effects of return flow, the concept of exposure time was introduced as an alternative to the residence time.

However, deep analysis of exposure time revealed unexpected complications that, in part, are still open questions.

The present contribution extends the concepts of water age and local flushing time to account for the return flow,

as for exposure time. These time scales are studied and compared to each other, aided by a modelling study

focusing on the Venice lagoon. Also, approximate methods to estimate the real time scales are introduced and

analysed. We show that the local flushing time well approximates, quantitatively, both water age and exposure

time in steady periodic hydrodynamic condition. Conversely, wind-driven advection makes exposure time to be

significantly different from water age and local flushing

time.

1 INTRODUCTION

Transport time scales are key parameters to assess water renewal of estuaries and tidal basins and they have long been used to this purpose.They have become more and more popular with the growing use of numer ical models, since renewal time scales are rational tools to “condense” huge amount of numerical data into intelligible, still quantitative, information (Deleersni jder and Delhez 2007). However, growing up is never straight forward: in a modern-day Tower of Babel, an ever-increasing confusion has involved terminology, use and estimation of time scales, despite valuable efforts towards a sounder theoretical framework (from Bolin and Rodhe 1973 to Delhez et al. 2014). This paper is a modest contribution to this sort of “climb ing” toward a well-established definition, and use, of renewal time scales.

The work is based on a numerical study that focuses on the Venice Lagoon (Italy), which is a tidally flushed, semi-enclosed basins with negligible freshwater inflow. We consider steady periodic hydro dynamic condition, driven by a sinusoidal tide imposed at the open sea. Under these hypotheses, water renewal within the lagoon of Venice is strongly controlled by diffusion. The inlets act alternatively as a sink and as a source, and a significant fraction of effluent water returns to the lagoon each flood tide. Accordingly, to correctly assess water renewal, a region larger than the Venice Lagoon itself has to be considered. To account for the effects of return flow, the concept of exposure time was introduced as an alternative to the residence time (Delhez et al. 2004). As for exposure time, we extend the concepts of water age and local flushing time to account for the return flow (Viero and Defina 2016). However, unexpected complications were found to affect the exposure time (Delhez 2013), and they also apply to water age and local flushing time as we define them (Viero and Defina 2016). From a mathematical point of view, the integral definition of these time scales leads to unbounded results. From a practical standpoint, when accounting for the return flow, time scales are shown to significantly depend on the size of the external domain (i.e. the size of the open sea included in the model). In other words, the (often arbitrary) location of the domain boundary affects the time scales computed within the basin. To overcome this problem, we introduce and analyse approximate methods to estimate the real time scales. Being aware that approximate time scales offer no counterpart to the detailed information contained in the full distribution of the real time scales, and their physical interpretation is also much weaker, we assess the accuracy of approximate estimates in comparison with the real time scales. We also show that the local flushing time well approximates, quantitatively, both water age and exposure time in steady periodic hydrodynamic condition. Conversely, wind-driven advection makes exposure

Figure 1. Bathymetry of the Venice Lagoon (meters above the mean sea level) based on the most recent bathymetric survey (2003). Dotted lines indicate the position of the divides between Lido, Malamocco, and Chioggia sub-basins. The white, dashed line denotes the open boundary of the computational domain (�).

time to be significantly different from water age and local flushing time.

2 MATERIAL AND METHODS

2.1 The Venice Lagoon

The Lagoon of Venice is a shallow, micro-tidal basin in the north-eastern Italy. It covers an area of ≈ 550 km^2 and the average water depth is ≈ 1.1 m. The three inlets of Lido, Malamocco, and Chioggia connect it to the Adriatic Sea (Fig. 1). The maximum tidal range is ≈ 1.5 m, with a dominant period of about 12 h. Fresh water inflow is neglected in the present analysis, since it is negligibly small compared to flow rates through the inlets. A very irregular bathymetry and morphology (Fig. 1) enhance diffusion processes within the lagoon. Further details can be found, e.g., in Solidoro et al. 2004, Cucco and Umgiesser 2006, Cucco et al. 2009, Umgiesser et al. 2014.

2.2 Numerical model

We used a semi-implicit, mixed Eulerian-Lagrangian finite elements numerical model to estimate hydrodynamics, transport and diffusion in the Venice Lagoon. The hydrodynamic module solves the 2D shallow water equations, modified in order to deal with flooding and drying processes in very irregular domains through refined sub-grid modelling of the bathymetry (Defina 2000, D'Alpaos and Defina 2007). Horizontal turbulent stresses are modelled by adopting the Boussinesq approximation; eddy viscosity is modelled according to the depth integrated model by Stansby (2003) and Uittenbogaard and van Vossen (2004). Details on the numerical implementation can be found in Defina (2003). The model has been validated in previous studies (D’Alpaos and Defina 2007, Carniello et al. 2011). The advection-diffusion module computes the fate of a conservative tracer by solving the depth averaged conservation equation where C = C(X, t) is the depth-averaged concentration, with X the horizontal position, U=U(X, t) the depth averaged velocity, and D h =D h (X, t) the twodimensional diffusivity tensor. In the model, diffusivity is assumed to be the same as the eddy viscosity computed by the hydrodynamic model. Details on the numerical approach and the results of an extensive validation can be found in Carniello et al. (2012) and in Carniello et al. (2014), respectively. The computational domain is made up of ≈ 51700 grid points (nodes) and ≈ 98500 triangular cells. The mesh includes the Venice Lagoon and a portion of the northernAdriatic Sea, connected to the lagoon through its three inlets (Fig. 1). In the following, we denote with ω the domain of interest (i.e. the Venice Lagoon) and with the extended domain that includes ω and a portion of Adriatic Sea (Fig. 1). We forced the hydrodynamic model by prescribing a sinusoidal tide of 0.5 m amplitude and a period of T = 12 h, typical of the northern Adriatic Sea (Cucco and Umgiesser 2006). In post-processing the model results to estimate renewal time scales, we assume that the time scale of phenomena at hand is much greater than the tidal period, thus referring to tidally-averaged concentration. This assumption may not be valid, e.g., close to the open boundaries of the control domain (i.e. close to the lagoon inlets). 2.3 Definition of renewal time scales 2.3.1 Residence time Considering a particle released at time t and location X= (x, y), the residence time, T r (X, t), is defined as the time the particle takes to reach the lagoon outlet for the first time (Zimmerman 1976). Takeoka (1984) extended this definition to cope with water parcel. Unfortunately, the spatial distribution of residence times can be achieved by running multiple numerical simulations, which require unacceptable computation time. Delhez et al. (2004) proposed a valuable approach, which allows estimating the mean residence time T r (X, t) everywhere within the basin, and for

every time t, by solving a single advection-diffusion

problem:

where δ ω is the characteristic function of the domain of interest, ω (i.e. the Venice Lagoon), defined as

Equation 2 has to be solved backward in time, with the reversed velocity field (i.e. U=-U), with initial condition $T_r(X) = 0$ and prescribing $T_r = 0$ at the lagoon inlets.

Alternatively, in the case of steady (periodic) flow conditions one can use the adjoint variable, C^*, introduced by Delhez et al. (2004), and solve the following problem

with initial condition $C^* = \delta_\omega$, again solved backward in time, with the reversed velocity field (i.e. U=-U), and prescribing $C^* = 0$ at the lagoon inlets.

The local, tidally-averaged adjoint variable, $\langle C^*(X, t)\rangle$, then decreases monotonically (backward in time) as a consequence of the tidal flushing, until it vanishes.

Under steady flow conditions, the residence time can finally be evaluated as

2.3.2 Exposure time

The exposure time of a particle, T_e, is defined as the time spent by the particle within the domain ω, irrespective of its temporary excursions out of ω (Monsen et al. 2002, Delhez 2006, Delhez 2013).

When a non-negligible fraction of water returns into the basin during each tidal cycle, as it is for the Lagoon of Venice, the exposure time has to be used in place of the “strict” residence time as previously defined, which is no longer an appropriate time scale.

To evaluate the exposure time, the computational domain ▯, which includes ω, must be sufficiently greater than ω. Actually, the size of should be infinite from the mathematical point of view. For a closed system, indeed, the concentration distribution tends to become uniform (not zero!) as time goes to infinity, regardless of the size of ▯, and the integral in Equation (5) diverges (Delhez 2013). In addition, the analytical solution to a simplified problem analysed by Delhez et al. (2004) (Eq. 17 therein) suggests that the exposure time tends to infinity also when advection is negligible compared to diffusion, irrespective of the size of the extended model domain, ▯.

According to Delhez (2013), in many studies where the exposure time is estimated using conservative tracers (e.g. Monsen et al. 2002, Delhez 2006), the integral in Equation 5 is found to converge only because it is implicitly assumed that the tracer leaving the domain through the open boundaries is permanently lost. This means that, when the return flow is considered, time scales significantly depend on the location of boundary conditions, which in most cases is arbitrarily chosen. Delhez (2013) proposed a solution in which a first order decay is imposed to the tracer. However, introducing a physically sound tracer decay and resolving the interconnected issue of boundary conditions (Delhez 2013) is far from being straightforward. One may ask if the very low concentrations of tracer that remain within the basin for a long time, and that considerably affect the average time scale, are actually significant in determining a renewal time scale. An answer to this point should also consider the purpose and the subsequent use of the computed time scales, e.g the physical/chemical/biological process at hand. A plain, frequently used solution is the use of approximate time scales, e.g., by fitting the modelled concentration decay with a simple analytical function (e.g. exponential) or to choose significant thresholds (e.g. 1/e ~ = 0.37). The number of applications of such procedures undoubtedly suggests the attractive power of these approaches. If we settle for approximate time scales, and particularly for the 1/e threshold criterion, complications affecting the exposure time reduce to carefully checking on the impact of boundary conditions prescribed at the open boundary of �, that is, one must make sure that is sufficiently large so that the prescribed boundary conditions negligibly affect the tracer concentration within the lagoon until the concentration reduces to 1/e. This point is further discussed in Viero and Defina (2016). Also, they showed that the 1/e threshold time scales poorly approximate the mean residence time when advection is significant (e.g. close to the inlets of tidal basins). 2.3.3 Local flushing time The local flushing time, T_f, is a renewal time scale that found several application in the literature (Chubarenko et al. 2004, Jouon et al. 2006, Sheng et al. 2009, Viero and Defina 2016). Plus et al. (2009) defined the local flushing time as "the time necessary for a significant portion of a water parcel to be replaced by water coming from outside the lagoon boundaries, i.e. from the ocean or the rivers". A number of different names have been used for very similar definitions: "residence time" (Cucco and Umgiesser 2006, Cucco et al. 2009), "e-folding time" (Wan et al. 2013, Jouon et al. 2006, Ouillon et al. 2010), "local residence time" (Abdelrhman 2002, Abdelrhman 2005), and "influence time" (Delhez et al. 2014). From the computational point of view, we numerically solve the advection-diffusion equation

with initial tracer concentration $C(X, t = 0) = \delta\ \omega$ and

with the condition $C(t) = 0$ prescribed at the open

boundaries. The local, tidally-averaged concentration,

$\langle C(x, t)\rangle$, of the tracer then decreases monotonically,

to finally vanish, as a consequence of tidal flushing.

Adopting the definition of "influence time" pro

posed by Delhez et al. 2014, the real local flushing time can be estimated at every position X within the lagoon as

An evident similarity can be found between the steady state definition of the residence time (Eq. 5) and the influence time. This similarity concerns in particu lar their complementary nature: both are computed solving an analogous problem but, contrary to the influence time, the residence time require backward in-time integration (along with the reversed velocity field). As it was pointed out by Delhez et al. 2014 and by Viero and Defina 2016, the local flushing time is not a residence time.

As previously mentioned, the return flow is not neg ligible for the lagoon of Venice. Therefore, to correctly assess the local flushing time, the open boundaries of the computational domain have to be set in the open sea, sufficiently far away from the lagoon inlets.

As for the exposure time, the integral estimate in Equation 7 comes to depend on the arbitrary location of the open boundaries, where the tracer is allowed to leave the domain but not to re-enter it. Again, as for the exposure time, to circumvent this problem we can resort to approximate estimations of the real time scale, thus defining the local flushing time as the time it takes for the local, tidally-averaged tracer concentration to decrease to 1/e (Chubarenko et al. 2004, Jouon et al. 2006, Plus et al. 2009, Sheng et al. 2009).

Viero and Defina 2016 showed that the 1/e threshold approximation in generally more reliable than the e-folding approach. Although providing sen sible underestimation of the time scales close to the inlets, where advection is non-negligible, the 1/e threshold approximation is progressively more accu rate for increasing time scales (i.e. where advection is negligible).

2.3.4 Water age

The concept of water age, T_a, has been assessed theoretically by Delhez et al. (1999), Deleersnijder et al. (2001), Deleersnijder et al. (2002), and Del hez and Deleersnijder (2002). The so-called CART (Constituent-oriented Age and Residence time The ory) deals with the age of water constituents such as dissolved salts, pollutants, plankton.The age of a water constituent is defined as “the time elapsed since a par ticle of a constituent of seawater left the region in which its age is prescribed to be zero” (Delhez et al. 1999). The age of pure water, that we deal with, can be seen as a sub-problem of this more general theoretical framework. Under steady periodic hydrodynamic conditions, water age $T_a(X)$ can be evaluated as the value of the tidally-averaged, instantaneous water age, $\langle a(X, t)\rangle$, which is approached as time evolves to infinity (Banas and Hickey

2005, Delhez et al. 2014). The instantaneous water age at $t = \infty$ can be found by solving (Thiele and Sarmiento 1990, England 1995) with initial condition $a(X, t) = 0$ everywhere, and by prescribing $a(X) = 0$ at the lagoon inlets. In Equation 8, the characteristic function of the domain, δ_{ω}, governs the natural ageing of water. It was noted by Delhez et al. (2004) that the problem governing the water age (Eq. 8) is analogous to that governing the residence time (Eq. 2), except for the need of the latter to be solved backward in time and with the reversed velocity field.Also, it was showed by Delhez et al. (2014) and by Viero and Defina (2016) that water age exactly corresponds to the local flushing time under steady (and steady periodic) hydrodynamic conditions. However, this definition of water age, in which the open sea is taken as the “zero-age region”, does not allow to account for the return flow, which in fact strongly affects the age distribution within the lagoon of Venice. A trivial solution to account for the return flow would be to set the region in which the age is prescribed to be zero far away from the inlets. Practically this entails solving Equation 8 in ▮, using δ as ageing factor and prescribing $a(t) = 0$ at the open boundary of ▮. By applying the “original” definition of water age, unfortunately, i) the water age accounts also for the time elapsed to travel from the “zero-age region” (i.e. from the open boundary) to the lagoon inlets, and not only for the time water spent within the lagoon, and ii) the greater the distance between the “zero-age region” and the inlets, the greater the age in the basin (Viero and Defina 2016). Due to this last point, and only in this sense, the water age is unbounded. Hence, the age within the lagoon comes to depend on the distance between the lagoon inlets and the boundary of where the condition $a(t) = 0$ is prescribed (i.e. on the arbitrary choice of the modeller). To overcome the first issue, while prescribing the boundary condition $a(t) = 0$ at the boundary of (i.e. far away from the lagoon inlets), we make the water to age only within the lagoon (i.e. we still use δ_{ω} in Eq. 8). To overcome the second issue, solutions can be adopted that are analogous to those proposed for the exposure time. However, the tidally-averaged water age, $\langle a(X, t)\rangle$, is a monotonically increasing function, which does not match with any significant threshold. Viero and Defina (2016) proposed to approximate the water age by applying the 1/e threshold on the ageing rate (i.e. on instantaneous age time derivative) rather than to water age itself. This approximate water age exactly corresponds to the local flushing time when considering steady

Figure 2. Return flow factor, defined as the fraction of

water

leaving during ebb that returns during flood, for the three

inlets of Lido, Malamocco, and Chioggia.

(periodic) flow conditions. Under time-varying con

ditions, these two time scales are substantially dif

ferent from each other: water age at time t_0 depends

on the past (i.e. $t \le t_0$), whereas the local flushing

time depends on the future (i.e. $t \ge t_0$) hydrodynamic

conditions.

3 RESULTS AND DISCUSSION

3.1 Return flow

First we focus on the role of the return flow. Following

Sanford et al. (1992), we define the return flow fac

tor, b, as the fraction of water leaving during ebb that

returns during flood. Thus we have $0 < b < 1$.

To quantitatively estimate the return flow factor, we

run the numerical model to solve for hydrodynamics

and advection-dispersion, prescribing a concentration

field $C = \delta_\omega$ as initial condition and $C(t) = 0$ at the

open boundary of the computational domain (white

dashed line in Fig. 1).

For each tidal cycle, we compute the volume of

tracer exiting and entering through each of the lagoon

inlets (Lido, Malamocco, and Chioggia). Two sim

ulations are run, starting from flood and ebb tide respectively, and the results are averaged to account for the effect of the initial tidal phase. As is shown in Figure 2, the fraction of tracer re-entering the lagoon increases with time as the concentration gra dient across the inlets reduces. Overall, a significant fraction of effluent water returns to the basin each flood tide; accordingly, to correctly assess water renewal in the lagoon of Venice, a region larger than the lagoon itself has to be considered (Viero and Defina 2016).

It has to be stressed that the effects of possible coastal currents, which may act to significantly reduce the return flow factor, are not considered in the present study.

3.2 Comparison of different time scales

To estimate each time scale, two different numerical simulations are run, starting from flood and ebb tide respectively. The time evolution of tracer concentration, computed at each node of the computational grid each time step, is post-processed to evaluate both integral and 1/e threshold renewal time scales. The results obtained from the two simulations are then averaged to achieve a measure of the mean behaviour of the basin, thus independent of the initial tidal phase. Figure 3 compares the different time scales defined in the previous Section. For the sake of simplicity, only the 1/e threshold approximations of the real time scales are here plotted. The accuracy of this approximation is discussed in the next Section 3.3. As already reported by Viero and Defina (2016), water age and local flushing time show an identical spatial distribution. Accordingly, both these time scales are depicted on a single map (right panel of Fig. 3). The spatial distribution is typical of diffusive, and non-advective, environments: water age increases quite monotonically from the inlets to the inner parts of the lagoon. The location of the divides between the three different sub-basins appears distinctly. The exposure time is very close to water age (and local flushing time) in magnitude, yet showing a quite different spatial pattern. We recall that, under steady periodic hydrodynamic condition, the only difference between exposure time and water age (or local flushing time) stems from advection. Actually, the modest Eulerian residual velocities resulting from the periodic hydrodynamics seem sufficient to explain the discrepancies between water age and residence time. Regions with the major residual velocities are located close to the inlets and to the major channels inside the lagoon, where inertial effects are able to produce asymmetry (i.e. residual advection) in the flow field. We carried out a further numerical experiment by superimposing to the same tide as previously a NE wind with a constant velocity of 10 m/s. Significant advection is added to the system, but the steady periodic flow conditions still hold. Water renewal is greatly enhanced (all the considered time scales greatly reduced) as a consequence of the wind action, as can be seen by comparing Figure 4 with Figure 3. Again, water age coincides with local flushing time. Conversely, residence time shows now a completely different pattern with respect to water age/local flushing time (Fig. 4). Clearly, this is due to the internal current that occurs from the Lido to the Chioggia inlet under the action of wind. 3.3 1/e threshold approximation of time scale The accuracy of the approximations previously introduced and used was partially assessed by Viero and Defina (2016). They analysed the typical concentration decays observed in different simulations at different locations within the Lagoon of Venice, and found similarities with the findings of Jouon et al. (2006). Here we report (Fig. 5) the spatial distribution of, respectively, the local flushing time (which is as well representative of both water age and exposure

Figure 3. Left panel: 1/e threshold exposure time computed through the adjoint approach. Right panel: 1/e threshold water

age, which exactly corresponds to the 1/e local flushing time.

Figure 4. Left panel: 1/e threshold exposure time computed through the adjoint approach. Right panel: 1/e threshold water

age, which exactly corresponds to the 1/e local flushing time.

time) computed using the integral approach of Equa tion 7 and its 1/e threshold approximation. We can observe that, for points located nearby the inlets where advection significantly affects the flow field, the 1/e threshold approximation substantially underestimates the mean time scale (particularly in terms of relative difference). For points located far from the inlets, where transport is dominated by diffusion, the mean time scale is accurately approximated using the 1/e threshold.

Figure 5. Left panel: local flushing time computed according to Equation 6. Right panel: 1/e threshold local flushing time.

We remark that, since the return flow is taken into account, the time scale provided by the integral approach varies with the size of ▯. On the contrary, the 1/e threshold local flushing time does not suffer the same dependence, since the computational domain we used is sufficiently large to affect the tracer concentra tion only after the 1/e threshold is reached in the most part of the lagoon.

4 CONCLUSIONS

Aided by a numerical study focusing on the Lagoon of Venice, we defined and assessed different time scales to describe water renewal, namely exposure time, water age, and local flushing time. Our reasoning thus concerned tidally flushed, semi-enclosed basins with negligible freshwater inflow, forced by steady periodic hydrodynamic conditions.

In the case of negligible coastal currents, the frac

tion of effluent water that returns into the basin each

flood tide was showed to be significant. Hence, the

return flow was considered in defining and estimat

ing all these time scales, as for the exposure time. To

this purpose, the definition of water age by Viero and

Defina (2016) was used.

Complications arise when the return flow is

accounted for in estimating renewal time scales

(Delhez et al. 2004, Delhez 2013). To overcome these

complications, we defined and used approximate time

scales. Their accuracy is shown to be acceptable in

regions located far away from the inlets where dif

Carniello, L., L. D'Alpaos, & A. Defina (2011). Modeling

wind waves and tidal flows in shallow micro-tidal basins.

Estuar. Coast. Shelf. S. 92, 263–276.

Carniello, L., A. Defina, & L. D'Alpaos (2012). Modeling

sand-mud transport induced by tidal currents and wind

waves in shallow microtidal basins: Application to the

venice lagoon (italy). Estuar. Coast. Shelf. S. 102–103,

105–115.

Carniello, L., S. Silvestri, M. Marani, A. DAlpaos, V. Volpe,

& A. Defina (2014). Sediment dynamics in shallow tidal

basins: In situ observations, satellite retrievals, and numer ical modeling in the venice lagoon. J. Geophys. Res. Earth 119, 802–815.

Chubarenko, B., V. Koutitonsky, R. Neves, & G. Umgiesser (2004). Modeling concepts. In I. Gönenç and J. Wolflin (Eds.), Coastal lagoons: ecosystem processes and modeling for sustainable use and development, pp. 231–306. Boca Raton: CRC Press.

Cucco, A. & G. Umgiesser (2006). Modeling the venice lagoon water residence time. Ecol. Model. 193, 34–51.

Cucco, A., G. Umgiesser, C. Ferrarin, A. Perilli, D. Canu, & C. Solidoro (2009). Eulerian and lagrangian transport time scales of a tidal active coastal basin. Ecol. Model. 220, 913–922.

D'Alpaos, L. & A. Defina (2007). Mathematical modeling of tidal hydrodynamics in shallow lagoons: A review of open issues and applications to the venice lagoon. Comput. Geosci. 33, 476–496.

Defina, A. (2000). Two dimensional shallow flow equations for partially dry areas. Water Resour. Res. 36(11), 3251–3264.

Defina, A. (2003). Numerical experiments on bar growth. Water Resour. Res. 39(4), 1092.

Deleersnijder, E., J. Campin, & E. Delhez (2001). The concept of age in marine modelling: I. theory and preliminary model results. J. Marine Syst. 28, 229–267.

Deleersnijder, E. & E. Delhez (2007). Timescaleand tracer

based methods for understanding the results of complex

marine models. Estuar. Coast. Shelf. S. 74, v–vii.

Deleersnijder, E., A. Mouchet, E. Delhez, & J.-M. Beckers

(2002). Transient behaviour of water ages in the world

ocean. Math. Comput. Model. 36, 121–127.

Delhez, E. (2006). Transient residence and exposure times.

Ocean Sci. 2, 1–9.

Delhez, E. (2013). On the concept of exposure time. Cont.

Shelf Res. 71, 27–36.

Delhez, E., J. Campin, A. Hirst, & E. Deleersnijder (1999).

Toward a general theory of the age in ocean modelling.

Ocean Model. 1, 17–27.

Delhez, E., B. de Brye, A. de Brauwere, & E. Deleersni

jder (2014). Residence time vs influence time. J. Marine

Syst. 132, 185–195.

Delhez, E. & E. Deleersnijder (2002). The concept of age in

marine modelling. ii. concentration distribution function

in the english channel and the north sea. J. Marine Syst.
31,

279–297.

Delhez, E., A. Heemink, & E. Deleersnijder (2004). Resi

dence time in a semi-enclosed domain from the solution

of an adjoint problem. Estuar. Coast. Shelf. S. 61, 691–702.

England, M. (1995). The age of water and ventilation timescales in a global ocean model. J. Phys. Oceanogr. 25,

2756–2777. Jouon, A., P. Douillet, S. Ouillon, & P. Fraunie (2006). Calculations of hydrodynamic time parameters in a semi-opened coastal zone using a 3d hydrodynamic model. Cont. Shelf Res. 26, 1395–1415. Monsen, N., J. Cloern, L. Lucas, & S. Monismith (2002). A comment on the use of flushing time, residence time, and age as transport time scales. Limnol. Oceanogr. 47(5), 1545–1553. Ouillon, S., P. Douillet, J. Lefebvre, R. Le Gendre, A. Jouon, P. Bonneton, J. Fernandez, C. Chevillon, O. Magand, J. Lefèvre, P. Le Hir, R. Laganier, F. Dumas, P. Marchesiello, A. Bel Madani, S. Andréfouët, J. Panché, & R. Fichez (2010). Circulation and suspended sediment transport in a coral reef lagoon: The south-west lagoon of new caledonia. Mar. Pollut. Bull. 61, 269–296. Plus, M., F. Dumas, J.-Y. Stanisière, & D. Maurer (2009). Hydrodynamic characterization of the arcachon bay, using model-derived descriptors. Cont. Shelf Res. 29 , 1008– 1013. Sanford, L., W. Boicourt, & S. Rives (1992). Model for estimating tidal flushing of small embayments. J.Waterw. Port C. ASCE 118, 635–654. Sheng, J., J. Zhao, & L. Zhai (2009). Examination of circulation, dispersion, and connectivity in lunenburg bay of nova scotia using a nested-grid circulation model. J. Marine Syst. 77(3), 350–365. Solidoro, C., D. Melaku Canu, A. Cucco, & G. Umgiesser (2004). A partition of the venice lagoon based on physical properties and analysis of general circulation. J. Marine Syst. 51, 147–160. Stansby, P. (2003). A mixing length model for shallow turbulent wakes. J. Fluid Mech. 495, 369–384. Takeoka, H. (1984). Fundamental concepts of exchange and transport time scales in a coastal sea. Cont. Shelf Res. 3(3), 322–326. Thiele, G. & J. Sarmiento (1990). Tracer dating and ocean ventilation. J. Geophys. Res. 95, 9377–9391. Uittenbogaard, R. & B. van Vossen (2004). Subgrid-scale model for quasi-2d turbulence in shallow water. In W. Uijttewaal and G. Jirka (Eds.), Shallow Flows. London: Taylor & Francis Group. Umgiesser, G., C. Ferrarin, A. Cucco, F. De Pascalis, D. Bellafiore, M. Ghezzo, & M. Bajo (2014). Comparative hydrodynamics of 10 mediterranean lagoons by means of numerical modeling. J. Geophys. Res. Oceans 119, 2212–2226. Viero, D. & A. Defina (2016). Water age, exposure time, and local flushing time in semi-enclosed, tidal basins with negligible freshwater inflow. J. Marine Syst. 156, 16–29. Wan, Y., C. Qiu, P. Doering, M. Ashton, D. Sun, & T. Coley (2013). Modeling residence time with a three-dimensional hydrodynamic model: Linkage with chlorophyll a in a subtropical estuary. Ecol. Model. 268, 93–102. Zimmerman, J. (1976). Mixing and flushing of tidal embayments in the western dutch wadden sea. part i: Distribution or salinity and calculation or mixing time scales. Neth. J. Sea Res. 10(2), 149–191. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Morphodynamic processes at Mariakerke: Numerical model implementation

R. Silva

Flanders Hydraulics Research, Belgium

Antea Group, Belgium

R. De Sutter

Antea Group, Belgium

R. Delgado

Flanders Hydraulics Research, Belgium

ABSTRACT: Within the framework of the project "Alternative maintenance measures for beach nourishment:

monitoring of a pilot site", the contribution of underwater nourishment to reduce maintenance, and thus extend

ing the lifetime of beach nourishment along the Belgium coast, without compromising coastal safety, is to be

assessed. This is based on an integrated approach, including field measurement campaigns, numerical modelling

and the definition and evaluation of a number of morphological indicators, for Mariakerke coastal area in Ostend,

Belgium. Within the first steps of the assessment was the implementation of a numerical model for the simulation

of the short term hydrodynamic and morphodynamic processes (beach response to storms and nourishments).

XBeach was implemented in its hydrostatic mode using hydrodynamic, sedimentary and morphological condi

tions measured, during a particularly stormy period, captured in one of the conducted field campaigns. Firstly, the

model was set-up in one-dimension. A sensitivity analysis and a calibration were performed. Then, it was set-up

in quasi-two-dimensions. The process of implementation, sensitivity analysis, calibration, testing, discussion

and main conclusions are here presented.

1 INTRODUCTION

1.1 Project overview

Within the framework of the Master Plan for Coastal Protection (Mertens et al., 2011) the Flemish Gov ernment aims to reinforce all “weak” coastal sections in order to meet the required safety levels to respond to a storm-event with a return-period of 1.000 years till the year 2050. A new concept was introduced that would later result in the Master Plan for the Flemish Bays (2014). The concept included a long-term vision requiring research on sustainable engineering and the creative use of soft coastal protection measures, as alternative to the traditional ones.

The project “Alternative maintenance measures for beach nourishment: monitoring of a pilot site” fits within this concept, aiming the assessment of the contribution of underwater nourishment to reduce maintenance, and thus extending the lifetime of beach nourishment along the Belgium coast, with out compromising coastal safety. A pilot shoreface nourishment was conducted at Mariakerke coastal area (Ostend). The coastal area has been intensively

monitored, in order to evaluate its efficiency and

sustainability, through regular campaigns carried out by Flanders Hydraulics Research, commissioned by Coastal Division 1 . The assessment consists in comparing the response of coastal sections, where beach nourishment alone has been conducted, to the one of adjacent coastal sections subjected to combined beach and shoreface nourishment. An integrated approach including the field campaigns, morphological modelling and the definition and evaluation of a number of morphological indicators was adopted. The approach requires the continuous collection and analysis of meteorological, hydrodynamic, sedimentologic and morphologic information throughout the duration of the project (2014–2018). 1.2 Study site overview The Belgian coastline is part of the Flemish coastal plain, a depositional system at the southern edge of the North Sea Basin. It comprises about 65 km of wave-exposed open coastline, between the more sheltered upper reaches of the Scheldt estuary in the east (the Netherlands) and Dunkerke (France) in the west 1 Maritieme Dienstverlening en Kust – afdeling Kust

(Figure 1). It is oriented SW-NE and consists of a

nearly continuous dune belt which constitutes the

primary natural defense of the low-lying hinterland

polder areas against flooding.

The tide along the coastline is asymmetric semi

diurnal. All beaches are situated in a macro-tidal

regime and the tidal range is typically between 3.5 m

at neap tide and 5 m at spring tide. This important

tidal range is linked to significant tidal currents, of

which the peaks generally slightly exceed 1 m/s in the

nearshore areas. Offshore waves are mainly driven

by westerly winds. Because of the shallow waters and the relatively short fetch, waves are typically short crested (Haerens et al., 2012). Fetches of more than 200 km can only be attained from the north, implying that storms from north-western to northern direction are the most severe (Van Lancker, 1999). The sediments on the shore are well sorted fine to medium sands. The natural sediments on the beach are characterized by fine to medium sand, mostly quartz grains, with a median diameter varying between 180 and 250 μm. In areas where beach nourishment took place, the grain size tends to be coarser, up to 400 μm (Haerens et al., 2012). A large part of all the 240 coastal sections are subjected to erosion of the beach and dune systems. Mariakerke coastal area is recognized as one of the “weak” coastal hot spots. There, the sea-dike is protruding with respect to the adjacent coastline, resulting in increased exposure to erosion.

Monitoring campaigns are being conducted, specif ically along sections 100 and 104 (Figure 2), aiming the measurement of water levels, waves, current pro files, turbidity, sediment concentration and sediment grain size near the bottom, and temperature, salin ity and sediment concentration in the water column. Measurements are regularly conducted:

- six times a year, for 13 hours, in a location 500 m offshore of section 104 over a mean depth of −7.5 m, temperature and salinity are measured along the water column (CTD) and water samples are collected;
- twice a year, for 6 weeks, multi-instrumental frames are deployed at two reference depths, −6.5 m and −3.5 m, along sections 100 and 104;
- on a less regular basis, for 8 to 24 hours, in the swash zone, using another multi-instrumental frame mea suring at the reference depth +1 m TAW, at sections 100 and 104.

The campaigns were initiated in September 2013.

In 2013 one six-week campaign and four 13-hours measurements were conducted.

During 2014, two six week campaigns, six 13-hours measurements and one swash zone campaign (pressure sensors only) were conducted. In June 2014 an Argus video-monitoring system was installed (by Deltares) at the top of one of the buildings located in-between sections 100 and 104. Regular topographic and bathy metric surveys have also been conducted at the pilot site. Figure 1. Overview of the Belgium coast and approximate location (yellow) of some of the Flemish banks monitoring stations (adapted from the Flemish coastal atlas online). Figure 2. Pilot site at Mariakerke coastal area and sections where monitoring campaigns are conducted. 1.3 Actions, interventions and campaigns The implementation of a morphological model for the simulation of short term beach response (to storms and nourishments) requires the knowledge of the forcing actions and of the corresponding beach response. Coherent sets of data were sought in 2013 and 2014. The hydrodynamic information (wave conditions and water levels) available from the Flemish banks monitoring network (accessed in 08/07/2015) was crossed with the surveys, the campaigns and the known interventions periods. In 2013 Akkaert-Zuid (AKZ), Kwintbank (KWI) and Raversijde (RAV) wave buoys (Figure 1) had a percentage of valid registers not inferior to 90%. In 2014 AKZ and KWI had a percentage of valid registers not inferior to 90%. In later December 2013 AKZ and RAV were not measuring. RAV did not register between April and September 2014, having an overall percentage of valid registers of 54%. Ostend tide gauge (Figure 1) registered 99.7% of the 5-min observations during 2013 and 91% during 2014. Storm periods were identified using the significant wave height (Hs) threshold for significant morphological change along the Belgium coast proposed by Haerens et al. (2012). Hs time series were plotted for 2013, 2014 (Figures 4, 5, respectively) and wave events with Hs >3.75 m indicated. Water level (WL) time series are also plotted

Figure 4. Significant wave height and water level data series, topo-hydrographic surveys and interventions in the 1 st (above) and 2 nd (below) semester of 2013.

and the events with WL >+5 m TAW indicated. The timings of the field campaigns, topo-hydrographical surveys and known interventions are incorporated as vertical bars. During 2013 (Figure 4) four storms were identified: in 11/09 (Hs,max = 4.0 m, AKZ); in 10/10 (Hs,max = 4.7 m, KWI); in 28/10 (Hs,max = 3.8 m, KWI); in 5-6/12 (Hs,max = 3.8 m, AKZ). Throughout the first half of 2013 the wave conditions were mod erate, whilst in the second half much more energetic, especially from September onward.

The storm of 5-6/12, the so-called Sinterklaas storm, is particularly well documented, not only the beach was surveyed before and after the storm, but also a field campaign was ongoing at section 104.

During this storm, the average peak period was 7.1 s (±2.5 s std) (AKZ). Exceptionally high water levels, in spring tides, were observed, reaching +6 m TAW. The highest significant wave height of the year occurred on October 10 th . Exceptionally high water levels, in spring tides, also occurred during this storm, reaching +5 m TAW.

During 2014 (Figures 5) two storms were identi fied: in 9-10/07 ($H_{s,max}$ = 4.3 m, AKZ); in 21-22/10 ($H_{s,max}$ = 4.6 m, KWI). During the first half of the year, the offshore wave heights were more intense than in the previous year, but without the occurrence of storms. The beach was surveyed before and after the field campaign, conducted under moderate wave conditions. Swash zone and offshore measurements were also made. The storm identified in 21-22/10 occurred during spring tide, with exceptional water levels exceeding +5 m TAW.

A field campaign was undergoing at sections 100

and 104, with all the instrumentation frames deployed.
Figure 5. Significant wave height and water level data series, topo-hydrographic surveys and interventions in the 1st (above) and 2nd (below) semester of 2014. A shoreface

survey had been conducted before the campaign and a beach survey during it. From what was presented, the most interesting periods for model implementation were: – The coincident with the occurrence of Sinterklaas storm in 12/2013, during which a field campaign was undergoing and pre storm and post storm beach surveys were conducted; – 03/04/2014 when the beach was surveyed before and after the field campaign, conducted for moderate wave conditions; – The coincident with the last field campaign, when a storm, occurred; however, even if the beach was surveyed afterward, it was not before the storm. 2 XBEACH REFERENCE SET-UP The period coincident with the occurrence of Sinterklaas storm was chosen for model set-up. 2.1 Before and after Sinterklaas storm profiles The pre storm profiles of sections 100 and 104 are a compositions of before storm measurements of the beach (RTK-GPS, 03/12/2013) and a single beam bathymetric survey (Jun-Jul/2013). After storm, the profiles are a composition of data extracted from a high resolution LIDAR beach survey (10/12/2013) and from a single beam bathymetric survey (20/05/2014). Figure 6 presents the profiles.

Figure 6. Measured profiles before and after Sinterklaas

storm at sections 100 and 104.

The mainly changes in the emerged beach were dune

erosion above circa +5 m TAW and slight accretion

below this elevation. This is consistent with the con

clusions from Trouw et al. (2015) that this was the

main effect of the storm.

The after storm profiles show movement of the exis

tent shoreface bar and, in the case of profile 104,

accretion. Since the post storm profiles result from

the composition of bathymetric surveys conducted 5-

7 months after the storm, these changes can result from

actions other than the storm. In fact, most of them are

likely due to the shoreface nourishment conducted in

Apr-May 2014 at sections 102-108.

2.2 Sinterklaas storm hydrodynamics

Wave conditions were measured at section 104 during the field campaign, over -6.5 m TAW, by one of the multi-instrumentation frames (Hercules). Water level was measured at Ostend tide gauge (Figure 7).

2.3 Reference set-up

XBeach (version Groundhog_Day) was set-up at sec tion 104 for the Sinterklaas storm period (04/12/2013 at 0:30 till 07/12/2014 at 23:30) in hydrostatic (insta tionary) surfbeat mode. In this mode, short wave variations on the wave group scale and the associated long waves are resolved. The total wave energy dissi pation (directionally integrated) due to wave breaking was modeled according to Roelvink (1993). In this formulation alpha is applied as wave dissipation coef ficient and gamma as breaker index. The total wave energy is calculated by integrating over the wave direc tional bins. Directional spreading was included, by defining a directional grid for short waves and rollers with bins (10 ∘) between -90 ∘ and 90 ∘ in the Carte sian convention w.r.t. the computational x-axis. This has a limited effect on the wave heights because of refraction, but it allows for obliquely incident waves and the resulting longshore currents (Roelvink et al.,
Figure 7. Wave conditions as measured during the field

campaigns over −6.5 m TAW at section 104 and water level as measured at Ostend tide gauge. 2010). Longshore gradients were ignored. At the offshore boundary, spectral conditions were parametrized using the campaign measurements. Absorbing-generating boundary conditions were activated, allowing any waves propagating perpendicularly towards the boundary to be absorbed and allowing for time-varying water level to be specified, as measured by Ostend tide gauge. Neumann boundaries (default) were adopted at lateral boundaries, this way there is locally no change in surface elevation and velocity. Simulations were performed with uniform sediments characterized by a median sediment diameter d 50 = 220 µm and a d 90 = 300 µm (default). The shoreline was considered to have a mean orientation of N55 ∘ E, resulting in 305 ∘ counter-clockwise w.r.t E at the local coordinate system. The simulation time was 96 hours, the approximate duration of the storm. 3 XBEACH ONE-DIMENSIONAL 3.1 One-dimensional set-up The computational grid was generated from the pre storm profile measured at section 104, using a Matlab toolbox made available by the model developers (http://oss.deltares.nl/web/xbeach/tools, accessed in 15/01/2015), which optimizes the CFL (Courant–Friedrichs–Lewy) criterion for the best model performance. The criterion is based on the low-frequency flow velocities in the non-linear shallow water equations. The result was a single gridline with 289 points and varying spacing (about 12m in the nearshore and 2m over the beach and dune). 3.2 XBeach parameters under analysis XBeach is controlled by the user with the help of a large number (~250) of model settings. These settings describe case-specific circumstances (~100), like boundary conditions and bed composition, but

Table 1. Adopted values for model parameters. minimum maximum

Parameter	default	allowed	allowed	WTI
gamma	0.55	0.4	0.9	0.541
facAs	0.1	0	1	0.123
fw	0	0	1	0
beta	0.1	0.05	0.3	0.138
alpha	1.0	0.5	2	1.260
wetslp	0.3	0.1	1	0.260

facSk 0.1 0 1 0.375

gammax 2 0.4 5 2.364

cf 0.003 0.001 0.0505 0.001

also the physical and the numerical behavior of the model (~150), like the wave form and the maximum wave height compared to the water depth. Settings to use with XBeach one-dimensional (1D), when cal culating dune erosion along the Dutch coast, were derived by comparing model results to a selection of 30 measurements from the laboratory and field exper iments. The optimization of 9 input parameters with a large influence on the calculations was done based on the erosion volume above Storm Surge Level (SSL) defining WTI ("Wettelijk Toetsinstrumentarium") set tings (van Geer et al., 2015).

A sensitivity analysis was performed for these same parameters, but for the conditions observed during Sinterklass storm at Mariakerke, at the Belgium coast. The parameters are mostly related to wave break ing processes, but also to morphology and the flow: gamma is the breaker parameter; facAs and facSk are calibration factors in time averaged flows, due to wave asymmetry and skewness, repectivelly; fw is the bed friction factor; beta is the breaker slope coefficient in the roller model; alpha is the wave dissipation coeffi cient; wetslp is a critical avalanching underwater slope; gammax is the maximum ratio wave height to water depth; and cf is the friction coefficient in flow. The settings were: 1) default; 2) minimum and maximum allowed values; 3) WTI. Table 1 presents the adopted values.

3.3 Results

The indicator of model performance selected for the sensitivity analysis was the erosion volume above SSL. It was defined as the 10 year return period level of +5.9 m TAW (IMDC/WL/KULeuven, 2004). The measured value of erosion volume above SSL was 17.82 m^3/m.

3.3.1 Default/WTI settings

Figure 8 presents the input pre-storm profile, the mea sured and the simulated post-storm profiles, and the corresponding difference between them, using default andWTI settings.The vertical lines limit the part of the profile which significantly changed (more than 1 cm) during the respective simulations. This active part of the profiles are presented in Figure 9. Figure 8. Simulated profiles using default andWTI settings and differences from measured post-storm profile. Figure 9. Simulated profiles using default andWTI settings and differences from measured post-storm profile. The estimated erosion above SSL was 14.26 m^3/m for default settings and 11.30 m^3/m for WTI, both lower than the measured. Both settings underpredict the width of the active profile (coincident for both), since significant changes are observed offshore of the estimated limit. Erosion above SSL is underpredicted for both settings, but immediately below this level erosion is overpredicted. This appears to be related to an overprediction of the dune slope. Neither of the settings reproduces the submerged bar at the post-storm profile. Understandable, since it is the result of the conducted shoreface nourishment. 3.3.2 Sensitivity to selected parameters Table 2 presents the simulation program defined for the sensitivity analysis and the corresponding erosion above SSL. Figure 10 presents the values of simulated erosion above SSL. The parameters are ordered by decreasing simulated range of erosion above SSL, showing their relative effect. Note that the simulation of the four referred settings for each parameter, does not guaranty that the maximum/ minimum erosion volumes had been found.

Table 2. Simulation program and obtained erosion above SSL in 1D implementation. SSL Erosion above

Simulation	gamma	facAs	fw	beta	alpha	wetslp	facSk	gammax	cf	(m3/m)
0 (default)	0.55	0.1	0	0.1	1.0	0.3	0.1	2.0	0.003	14.26
1 (gamma)	0.4	0.1	0	0.1	1.0	0.3	0.1	2.0	0.003	6.31
2 (gamma)	0.541	0.1	0	0.1	1.0	0.3	0.1	2.0	0.003	13.71
3 (gamma)	0.9	0.1	0	0.1	1.0	0.3	0.1	2.0	0.003	53.55
4 (facAs)	0.55	0	0	0.1	1.0	0.3	0.1	2.0	0.003	16.90
5 (facAs)	0.55	0.123	0	0.1	1.0	0.3	0.1	2.0	0.003	13.77
6 (facAs)	0.55	1	0	0.1	1.0	0.3	0.1	2.0	0.003	-6.23
7 (fw)	0.55	0.1	1	0.1	1.0	0.3	0.1	2.0	0.003	1.70
8 (beta)	0.55	0.1	0	0.05	1.0	0.3	0.1	2.0	0.003	26.79
9 (beta)	0.55	0.1	0	0.138	1.0	0.3	0.1	2.0	0.003	11.42
10 (beta)	0.55	0.1	0	0.3	1.0	0.3	0.1	2.0	0.003	8.10
11 (alpha)	0.55	0.1	0	0.1	0.5	0.3	0.1	2.0	0.003	23.78
12 (alpha)	0.55	0.1	0	0.1	1.260	0.3	0.1	2.0	0.003	13.31
13 (alpha)	0.55	0.1	0	0.1	2.0	0.3	0.1	2.0	0.003	10.98

14 (wetslp) 0.55 0.1 0 0.1 1.0 0.1 0.1 2.0 0.003 22.02

15 (wetslp) 0.55 0.1 0 0.1 1.0 0.260 0.1 2.0 0.003 14.88

16 (wetslp) 0.55 0.1 0 0.1 1.0 1.0 0.1 2.0 0.003 2.86

17 (facSk) 0.55 0.1 0 0.1 1.0 0.3 0 2.0 0.003 16.03

18 (facSk) 0.55 0.1 0 0.1 1.0 0.3 0.375 2.0 0.003 10.61

19 (facSk) 0.55 0.1 0 0.1 1.0 0.3 1.0 2.0 0.003 4.98

20 (gammax) 0.55 0.1 0 0.1 1.0 0.3 0.1 0.4 0.003 1.10

21 (gammax) 0.55 0.1 0 0.1 1.0 0.3 0.1 2.364 0.003 14.52

22 (gammax) 0.55 0.1 0 0.1 1.0 0.3 0.1 5.0 0.003 16.26

23 (cf) 0.55 0.1 0 0.1 1.0 0.3 0.1 2.0 0.001 23.45

24 (cf) 0.55 0.1 0 0.1 1.0 0.3 0.1 2.0 0.0505 10.23

25 (WTI) 0.541 0.123 0 0.138 1.0 0.260 0.375 2.364 0.001 11.30

Figure 10. Limits of variation of simulated erosion above SSL for varying parameters and values for default and WTI settings.

Due to nonlinearities the occurrence of local max ima/minima in between is possible. Also, a change of a parameter setting might have a bigger effect in combination with other parameter setting.

Figure 11 presents the simulated profiles for vary ing wetslp, the simulated profile using default settings (used as reference) and their differences. Only the part of the profiles between the offshore limit of sig nificant changes (for default settings) and the shore are shown. wetslp is a morphological parameter that accounts for the slumping of sandy material from the dune face to the foreshore during storm-induced ero sion. Increasing it corresponds to increase the build-up capacity of sand, thus reducing berm erosion and con sequent accumulation of sand immediately below this level.The erosion volumes above SSL for wetslp = 0.1, 0.260 (WTI), 0.3 (default) and 1 were, respectively,

Figure 11. Simulated profiles for varying wetslp (S14, S15, S0, S16) and differences from measured profile post-storm profile.

22.02, 14.88, 14.26, 2.86 m^3/m. Similar results were produced for each of the parameters. The knowledge of the model response to the varying parameters allows for the model calibration for the set-up conditions.

3.4 Calibration

Starting from the comparison of simulations for default andWTI settings (Figures 10, 11). An apparent overprediction of the dune slope (similar to the initial slope) was noticed for both settings. Also, default set tings resulted in a profile shape closer to the observed thanWTI, at the wet beach.The calibration was initiated using default settings and tuning parameters related with slope stability.

3.4.1 wetslp

The sensitivity tests confirmed that the dune slope is very sensitive to wetslp. Considering the profile shape and erosion above SSL, as indicators of the model performance, a value wetslp = 0.1 was selected. For this value, erosion above SSL is not the closest to the measured, but the dune slope is. However, an underes timation of volume accumulation in the dune toe was still observed.

3.4.2 dryslp

Avalanching is introduced via a critical bed slope for both the wet and dry areas (wetslp and dryslp). It is considered that inundated areas are much more prone to slumping and therefore two separate critical slopes are used. A calibration of dryslp was also attempted. dryslp = 0.1, 0.15, 0.2, 0.3, 0.4, 0.5, 1 (default), 2 was considered.The respective erosion volumes above SSL were 25.57, 23.01, 22.52, 22.34, 22.12, 22.02, 22.02, 22.02 m^3/m. The effect of varying this parameter was less pronounced than that of wetslp and more localized in the dry beach. Increasing dryslp also corresponds to increasing the build-up capacity of sand, but in the dry beach, thus reducing berm erosion in the upper part of the dune. The value selected for this parameter was dryslp = 0.2.

3.4.3 facSk

The excess of erosion observed in the dune toe was then calibrated through facSk. The considered values were facSk = 0.1 (default), 0.2, 0.3, 0.375 (WTI), 1 (Figure 12). The erosion volumes above SSL were 22.52, 20.98, 19.49, 18.17, 10.45 m^3/m. Increasing facSk results in less changes in the profile (slight less accumulation in the wet beach and erosion in the beach berm). The value for this parameter that resulted in profile shape and erosion above SSL closer to the measured was facSk = 0.3.

3.4.4 Default/WTI settings/Calibration

Figure 13 presents the simulation results for default, WTI and calibrated settings. The erosion above SSL simulated through calibrated settings (19.49 m 3 /m) is similar to the measured (17.82 m 3 /m). The profile shape in the wet beach and dune is very close to the measured profile.

4 XBEACH QUASI-TWO-DIMENSIONAL

4.1 Two-dimensional set-up

For quasi-two-dimensional simulations (Q2D) the model was set-up as indicated in Section 2.3. The 2D computational grid was generated using the same tool box than in the 1D case, by replicating alongshore the pre storm profile measured at section 104. Figure 12. Simulated profiles for varying facSk and differences from measured profile post-storm profile. Figure 13. Simulated profiles using default, WTI and calibrated settings and differences from measured post-storm profile. It had 288 points cross-shore and 65 points alongshore, with a cross-shore spacing of approx. 12 m in the nearshore and 2 m over the beach and dune and an alongshore spacing of approx. 5 m in the center and 20 m near the lateral boundaries (Figure 14). 4.2 XBeach aspects under analysis The aspects inspected with Q2D simulations were: – Simulations comparison for default and WTI settings (following the comparison performed in 1D) and comparison with 1D simulations; – Effect of varying parameters which could be connected with simulation time (dtbc, tintm, number of processors); – Effect of deepening the profile to -20m offshore; – Effect of changing wave grid resolution; –

Effect of changing spatial grid resolution, both in cross-shore and alongshore direction.

Figure 14. Bathymetric input grid in Q2D simulations.

4.3 Results

The selected indicators of model performance were the same as in 1D simulations: erosion above SSL (+5.9m TAW) and the overall active profile changes, evalu ated at the central profile of the Q2D grid. For each aspect under analysis, the simulated profiles were plot ted together with the measured post storm profile and the differences between them.

4.3.1 Default/WTI settings

The results for default and WTI settings were similar to the verified in 1D simulations. For both settings, erosion above SSL was underpredicted, but immedi ately below this level, erosion was overpredicted. The erosion above SSL was, respectively, 10.90 for default settings and 7.40 m^3/m for WTI (in 1D simulations it was 14.24 m^3/m and 12.43 m^3/m, respectively). The simulation time was 92 and 94 h (on two processors).

4.3.2 1D/Q2D

Differences between Q2D and 1D simulations (Figure 15) are smaller in the case of default settings and more significant in the case ofWTI settings, especially in the dune front. In both cases, erosion volumes above SSL are higher in 1D simulations.

For default settings, 14.24 m 3 /m compared to

10.90 m 3 /m in Q2D simulations. For WTI, the differ

ence is higher, 12.43 m 3 /m compared to 7.40 m 3 /m.

The simulation time is significantly lower in 1D (~2 h)

than in Q2D (~90 h, real time storm duration 96 h). A

local PC Q2D simulation (WTI) was carried out. Ero

sion above SSL was the same (7.40 m 3 /m), but the

simulation time quite higher (218 h).

4.3.3 Parameters which could be related with simulation time

Q2D simulations revealed time costly. In a tentative

to reduce this time, different settings were tested and

their effects evaluated. The time step used to describe

time series of wave energy and long wave flux at off

shore boundary (dtbc) and the time interval for output

of mean variables (tintm). Besides these, the number of

processes in parallel simulations was analyzed. It had
Figure 15. Q2D and 1D simulated profiles using default settings and differences from measured post-storm profile.
been noticed, during 1D simulations, that the number of processes significantly affected the results. Therefore, 1D simulations were conducted with the same processes matrix (on 2 processors). No significant differences were found in the results for dtbc equal to 0.05 s and 1 s. Erosion volumes above SSL were 10.90 and 10.91 m 3 /m, respectively, and only a slight reduction of simulation time from 92 to 87 h was achieved. Also, no significant differences were found by changing tintm in the latest simulation, from 342000 s to 3600 s. Only a slight increase in the simulation time to 94 h, eventually due to the increase of the amount of information to process between outputs. Contrary to what had been observed in 1D simulations, the number of execution processors did not affect the results. Erosion volumes above SSL were 10.91, 10.84, 10.86, 10.86 m 3 /m, respectively, for 2, 10, 12, 20

execution processors. On the other hand, a strong reduction in simulation time was found 87, 21, 18, 12 h, respectively. 4.3.4 Deepen profile The toolbox used for the grid generation deepens the profile at the offshore boundary to −20 m (by default), in order to properly allow the application of the Longuet-Higgins and Stewart-type bound infragravity waves. The artifact aims the fulfillment of a criterion for the ratio between the wave group velocity and the wave celerity (dependent on the wave period), which avoids the generation of unrealistic big long waves at the boundary. Even though, at the peak of the storm, when larger wave periods occur, this criterion was exceeded. In these cases, the incident long wave amplitude is artificially reduced at the boundary by XBeach. The effect of the profile deepening was evaluated for default and WTI settings. It was found to be minor, at the active part of the profiles. The erosion volumes above SSL reduced from 10.91 to 10.80 m^3/m for default settings and from 7.71 to 7.42 m^3/m for WTI.

Figure 16. Simulated profiles for varying directional wave grid resolution 10°, 22.5° and 45° and differences from measured post-storm profile.

Figure 17. Simulated profiles for varying cross-shore spatial grid resolution dxmin = 2, 5, 10 m and differences from measured post-storm profile.

4.3.5 Wave grid

The effect of varying wave grid resolution in the offshore boundary (10°, 22.5°, 45°) was inspected. The erosion volumes above SSL were, respectively, 7.43, 9.72, 12.89 m^3/m. The simulation times were, respectively, 12, 8, 7 h. Figure 16 presents the results for the active profile. These show an important effect from the directional wave grid resolution. In general, for the tested conditions, decreasing resolution causes enhanced erosion and decreasing simulation time.

4.3.6 Computational grid

The Q2D bathymetric grid had varying cross-shore and

alongshore spacing. The effect of the minimum spac

ing (dxmin and dymin, respectively) was evaluated.

Figure 17 presents the simulated profiles for varying

cross-shore resolution, dxmin = 2, 5, 10 m, showing a

decrease in dune erosion and accumulation in the dune

base for decreasing resolution.This effect is very small

between the 5 m and the 10 m tests. Figure 18. Q2D simulated profiles for default, WTI and 1D calibration settings. Accordingly, erosion above SSL should be similar for these tests and lower than in the 2 m. However, it is not (7.43, 1.95, 14.35 m 3 /m, respectively). This happens, because in the 10 m test additional erosion occurs above the dune. In fact, this is not erosion. The difference is already present in the input profile, which in the 10 m case is flattened above the dune, due to interpolation.As expected, the simulation time reduces with resolution decrease, 12, 7, 6 h, respectively. Varying alongshore resolution, dymin = 5, 10, 15 m, resulted in identical profiles and erosion volumes above SSL (7.43, 7.38, 7.54 m 3 /m). For the simulated conditions, the resolution of the spatial grid had a major influence in the simulation results, but only in the cross-shore direction. 4.4 Applying the calibration parameters derived from the 1D model The settings resulting from the 1D calibration, were applied in a Q2D simulation, for the same conditions. The results are presented in Figure 18 together with the obtained with default and WTI settings. The calibrated settings give the profile closest to the measured. Erosion volume above SSL was 13.90 m 3 /m. 5 CONCLUSIONS XBeach was set-up and calibrated at section 104 of the Belgium coast (Mariakerke, Ostend). The forcing and calibration conditions were high energetic as measured during Sinterklaas storm in December 2013. During the event, a field campaign was ongoing and pre storm and post storm beach surveys had been conducted. Maximum significant wave heights reached 3.8 m offshore and the average peak period was 7.1 s. Exceptionally high water levels occurred, reaching +6 m TAW. The measured erosion volume above SSL

(+5.9 m TAW) was 17.82 m 3 /m. XBeach was firstly set-up in 1D hydrostatic mode. Model performance was investigated using as indicators the profile shape and the erosion volume above

SSL. A sensitivity analysis was performed to 9 input

parameters with a large influence on the calcula

tions. A rank for their influence was presented as a

function of the simulated range of erosion volume

above SSL (gamma, facAs, wetslp, beta, gammax, cf,

alpha, fw, facSk). The performance of the model was

assessed for default and WTI settings (derived in van

Geer et al., 2015). In the wet beach, default settings

resulted in a profile shape closer to the observed, whilst

WTI settings tended to overpredict accumulation. For

both settings, erosion above SSL was underpredicted,

but immediately below this level overpredicted. The

knowledge of the model response to the varying

parameters allowed for its calibration, using param

eters that could influence dune slope. Tunning wetslp,

dryslp and facSk allowed the achievement of a sim

ulated profile very close to the measured post storm

profile and of an identical erosion volume above SSL.

Afterwards, XBeach was set-up in Q2D (alongshore

uniform) hydrostatic mode for the same conditions.

The comparison of 1D and Q2D simulations showed

that there is a tendency of less dune erosion and less

wet beach accumulation in Q2D.This is consistent with the findings in Lanckriet et al. 2015, that XBeach in 1D has a more energetic long wave band than an equiv alent model in Q2D because the 1D model is unable to resolve wave directional spreading in the long wave band. However the reduction is settings dependent. In the case of default settings differences are less sig nificant than in the case of WTI settings. Similarly to what was verified in 1D, default settings resulted in a profile shape closer to the observed in the wet beach. For both settings, dune slope was overpredicted. A better model performance was achieved by applying the settings resulting from 1D calibration, and erosion volume above SSL approached the measured.

Q2D simulations revealed time costly (almost real time). In trying to reduce simulation time, different settings were tested, but with an insignificant effect in the results and a minor effect in simulation time. The simulation time was greatly reduced by increas ing the number of execution processors. Contrary to what had been observed in 1D case, no significant differences were found in the results. This should be related to the fact that when running XBeach in par allel mode, the model domain is subdivided and each of these is then computed on a separate core. Since no information about slopes is exchanged over the domains boundaries, the avalanching algorithm does not function there.An inadequate choice of boundaries might return inadequate results. In the 1D simulations the automatic subdivision could have split the pro file in an active zone. Whilst, in 2D this could have not happened. Parallel implementation of XBeach was recently improved (RISK-KIT 2015).

For the tested set-up, the deepening of the profile offshore, to optimize the model performance has only

Design features of the upcoming Coastal and Ocean Basin in Ostend, Belgium

A. Kortenhaus, P. Troch, N. Silin, V. Nelko, P. Devriese, V. Stratigaki & J. De Maeyer

Department of Civil Engineering, Ghent University, Zwijnaarde (Ghent), Belgium

J. Monbaliu, E. Toorman & P. Rauwoens

Department of Civil Engineering, KU Leuven, Leuven, Belgium

D. Vanneste, T. Suzuki, T. Van Oyen & T. Verwaest

Flanders Hydraulics Research, Antwerp, Belgium

ABSTRACT: The new Coastal and Ocean Basin (COB) at the Greenbridge Science Park in Ostend, Belgium

is under design. The laboratory will provide a versatile facility that will make a wide range of testing possible,

including the ability to generate waves in combination with currents and wind at a large range of model scales.

The facility is part of the Gen4Wave project on offshore renewable energy and coastal engineering in Flanders,

Belgium. The COB will allow users to conduct tests for

coastal and offshore engineering projects as well as for research. The basin will have state-of-the-art generating and absorbing wavemakers, a recirculation current system, and a wind generator. It will be possible to generate wave-current interactions in the same, opposite and oblique directions. The basin is expected to be operational in 2017. This paper presents an overview of the basin’s capabilities, the ongoing work, and some results of the design work.

1 INTRODUCTION

A new Coastal and Ocean Basin (COB) to be built in the municipality of Ostend in Belgium, within the context of the Gen4Wave project, is under design. The COB will come to satisfy the demands from not only the academic sector, but also from private companies developing coastal and offshore technology. The facility will cover a wide range of needs while keeping the lowest possible operating costs, which has led to the adoption of some unique solutions both in the management of the project and in the engineering solutions.

Scientific research in the fields of coastal and offshore engineering, like many other engineering domains, has evolved into a so-called integrated research methodology, combining both numerical modeling and physical scale model testing. In this context the scarcity and limitations of existing testing infrastructure for maritime research and development calls for further investment in this sector.As part of the credentials for the financing of the Gen4Wave project, the interest in the project by both the private sector and the academic sector, had to be demonstrated. This was successfully achieved receiving ample expressions of interest for the commercial use of the facility, ensuring the complete allotment of available testing times for the private sector during the initial phase of operation.

The three partners in the COB project, i.e.

Ghent University (UGent), University of Leuven

(KU Leuven), and Flanders Hydraulics Research (WL by its acronym in Dutch), have a longstanding research record in projects involving wave propagation, coastal processes, and coastal structures. Swift access to a large-scale facility with multi-directional wave, currents and wind, will enable breakthrough experimental research in the field of coastal engineering. Flanders has a long tradition in coastal engineering supported by the infrastructure of WL in Borgerhout. However, the dimensions of the wave tank of WL (17.5 m × 12.2 m × 0.45 m) are limited and only longcrested waves can be generated so that it is more suitable for coastal engineering applications. Therefore, the proposed COB project complements the existing infrastructure not only for coastal engineering applications but also for offshore as well as wave and tidal energy applications. In the field of renewable energies further understanding of an optimal wave energy converter (WEC) farm layout for realistic 3D wave-current fields will be pursued. This comprises the production of a data set to validate numerical models of wave energy converters, including mooring effects. In the same area experimental research towards numerical model validation of wave slamming on complex floating objects, such as (but not limited to) WECs, is planned. Fundamental questions still remain on the impact loading of structures due to combined waves and currents, and the consequent structural response. Examples of this are the prediction of wave overtopping at harbour quay walls and the interaction of overtopping flows with storm surge walls, the prediction of damage at scour protections for monopile wind tur bines and the impact of vegetation on the bottom erosion parameters in combined wave-current con ditions. These research questions will be tackled in the COB, amongst others, and will allow coastal and offshore engineering research in Flanders on a high international level. To this end, swift access to a large scale facility with multi-directional wave and current generation is indispensable.

Finally, the COB will enable further studies of the role of wave-current and wave-wave interactions on the excitation of freak waves. This research line has been seriously hampered by the scarcity of 3D wave basins capable of generating high quality flows for wave-current interaction studies.

2 BACKGROUND

As the project has gathered interest from both industry and academia, it received financial support from dif ferent organizations. UGent together with KU Leuven obtained financial support for the wave-current gen eration system from the Hercules Foundation in the amount of approximately 2.3 million Euros. UGent and KU Leuven provide complementary funds in the frame of this funding. The Hercules Foundation is a structural funding instrument for investment in scale infrastructure for fundamental and strategic research in scientific disciplines.

The Gen4Wave project has made available the resources to acquire additional relevant equipment to have the COB fully operational. The Gen4Wave project is financed by the Agency for Innovation by Science and Technology (IWT), which is pro viding approximately 3 million Euros for the initial investments, and 2 million Euros for the operational phase.

The building and main infrastructure will be pro vided by the Maritime Access Division of the Flemish Ministry of Mobility and Public Works. This includes the housing and the concrete wave tank structure.

2.1 Gen4Wave project

The COB is part of the Gen4Wave action plan result ing from an initiative of the Flemish government that supports the innovative manufacturing industry in the field of renewables. The plan has been developed by Agoria Renewable Energy Club (AREC) and UGent upon request of Generaties, a working group posi tioning Flanders in the field of renewable energy production, and came to existence thanks to the consor tium UGent (coordinator), KU Leuven and Flanders

Hydraulics Research (WL). The Gen4Wave project is defined by three pillars:

• Gen4Wave Coastal and Ocean Basin (COB): The first pillar is the infrastructure that will allow users to perform model tests of coastal structures. The infrastructure is versatile and has been designed keeping in mind a wide range of applications. • Gen4Wave R&D Projects: The second pillar is the support of innovation projects in line with the Gen4Wave R&D Roadmap. • Gen4Wave Energy Platform: The third pillar is a platform for dialogue, bringing together the academic institutions and the industry, namely the Gen4Wave Energy Platform. This platform works as a dynamic breeding ground for new projects. The demand for physical model testing has led to a shortage of available capacity in Europe. In preparation for the COB proposal, we further conducted an exhaustive study of the existing infrastructure in Europe. The first step was to see how COB positions itself against other testing facilities in Europe in terms of dimensions and capabilities. In that respect, the Unique Selling Propositions (USPs) of the COB are: 1. COB-dimensions allow a wide range of applications at various scales at an acceptable cost. 2. Ability to accurately reproduce (combined) loads from waves, current and wind. 3. Powerful machinery able to generate significant flow loads. 4. High directionality, with currents and waves interacting at random directions relative to each other. A unique opportunity to create a research synergy was found between the COB and the new towing tank of the WL in Ostend, both infrastructures sharing the same location in Ostend. 2.2 Synergy with towing tank Part of the expansion of the activities of the WL is the building of a new towing tank. By having both initiatives take place in the same location, a physical modelling cluster will result, reinforcing both initiatives. This cluster will allow for synergy in aspects such as the sharing of expertise, resources and administration of the infrastructure. The infrastructure is illustrated in Figure 1. The red section is the COB, and the blue section is the towing tank by the Hydraulics Laboratory. 2.3 Consortium The consortium is composed by Ghent University, the Catholic University of Leuven and Flanders Hydraulics Research. From Ghent University the following research groups participate: (1) Coastal

Engineering research group, (2) Maritime Technology Division and (3) the research group of Mechanics of Materials and Structures. The Catholic University of Leuven participates with the Hydraulics Laboratory. These groups will contribute with their specific expertise in the upcoming academic projects in the COB: coastal and offshore structures engineering, composite materials, slamming and deformation, wave and tidal energy, maritime technology, wave-current and wave-wave interaction and sediment mechanics. The organization of the consortium is stated in a consortium agreement, defining the contributions of

Figure 1. Location of the upcoming research cluster near the port of the Ostend (red: COB Gen4Wave, blue: towing tank operated by Flanders Hydraulics Research, green: shared office space).

each partner and the joint activities to be pursued. This involves also a management committee to exploit the COB on a daily basis.

3 DESIGN BASIS

In this section the specific elements that are part of the Gen4Wave Coastal & Ocean Basin (COB) are reviewed in the context of the COB targeted function ality. This comprises the basin facilities (section 3.1), the infrastructure to generate loads (waves, currents and wind) and water levels, see sections 3.2 to 3.6, and the instrumentation and data acquisition (section 3.7 and 3.8).

3.1 Location

The COB will be located at the Green Bridge Science Park close to the port of Ostend (Figure 1). The Sci ence Park covers about eighteen hectares and provides

the necessary space and services for the planned wave

basin as well as for the development of future industrial Figure 2. Overview of the COB facility. activities.The science park also houses the eponymous incubator Green Bridge. This can be used by startup companies that are associated to the wave basin to develop commercial activities thus having the opportunity to grow. The proximity to the port of Ostend is an advantage as the site is a strong renewable energy hub. 3.2 Basin facilities The COB lab (52 m × 42 m) will host the basin (30 m × 30 m) and several autonomous systems that allow to achieve its whole capability (Figure 2). The main systems are the wave and current system, the wind generator and the water transfer system which are connected to the data acquisition system (DAQ) to be able to rapidly set up testing scenarios and to collect the main testing parameters. There are also other systems that are needed to efficiently operate the COB, most notably the bridge crane, the carriage (or access bridge), the forklift and the wheel loader. These devices will be described in the following. An overhead crane with a suitable capacity (10tons) will be available to displace heavy items in and out of the wave tank (i.e. scaled models, structures, equipment, wind generator, etc.). The crane will cover the entire area of the COB lab. It will be possible to access the wave basin with an electric forklift or a wheel loader making the model construction process as simple as practically possible. The operation of the basin will be possible from two control locations. Furthermore, the COB will be equipped with an access bridge. This is a mobile structure which allows the users to reach every location or instrument in the basin without having to go inside the water. These “dry” conditions facilitate the work of the experimentalists. Also, the measuring bridge provides a close view of the experiments. Working with models and operating such infrastructure also requires a technical working space where things can be assembled, processed, manipulated and repaired. For this purpose there is a workshop area immediately next to the COB lab. Furthermore, a range of materials will be stored for the construction of models (stones in a variety of sizes, model blocks, guiding walls, etc.). The synergy with the towing tank will allow co-operation in specialized tasks.

Table 1. Selected examples 1 of existing coastal wave basins. Maximum Dimensions wave

Name (L ×W × D) (m) height (m)

Portaferry (Ireland) 18 × 16 × 0.65 0.55

DHI (shallow basin, 25 × 35 × 0.8 est. 0.4

Denmark)

Aalborg basin 1 15.7 × 8.5 × 0.75 0.2

(Denmark)

Aalborg basin 2 12 × 17.8 × 1 est. 0.5

(Denmark)

Delta basin 50 × 50 × 1 0.45

(Deltares, Netherlands)

Pacific basin 22.5 × 30 × 1 0.4

(Deltares, Netherlands)

Atlantic basin 75 × 8.7 × 1 0.45

(Deltares, Netherlands)

Tsunami wave basin 48.8 × 26.5 × 1.37 0.75

(Oregon, USA)

Coastal basin 15.5 × 10 × 0.5 0.3

(Plymouth, UK)

HRWallingford (UK) 27 × 55 × 0.8 0.25

1)The list of coastal wave basins world-wide has been limited

to a number of key facilities.

3.3 Wavemaker

The wavemaker is the most important mechanical sys

tem in the COB. The wavemaker will be purchased

through an international tender and therefore a lot of care has been taken to produce the right specifications. The first analysis towards the determination of these specifications involved the analysis of typical mod elling scenarios. Based on these results extra margins in some of the parameters were defined to allow for further flexibility or capability.

The COB wavemaker will ideally cover two sides of the basin, forming a corner. This setup allows for a larger range of oblique wave angles. Also, as the current can be reversed in direction, any relative angle between the current and the waves can be achieved.

The COB will be able to cover test conditions from coastal to near offshore. In Table 1 some examples of existing coastal wave basins are shown. The wave height in these basins is often limited by the available water depth and this is often related to the horizontal dimensions of the basin for most of the 3D coastal models. On the other hand, too large coastal models would increase the construction and operation costs of these models. The COB will therefore allow coastal models in up to 1.1 m water depth and a maximum depth-limited wave height of 0.55 m (regular waves).

In a similar way the COB will offer additional off shore modelling capability. Offshore tests often cover a very wide range of conditions (see e.g. the selected offshore test basins in Table 2). In the case of the COB

the most relevant offshore application will be the tests

of wave energy converters and therefore a compromise

water depth of 1.4 m was adopted adding a central pit

in excess of 4 m depth for mooring applications. Table 2. Selected examples of offshore test basins. Maximum Dimensions wave Name (L ×W × D) (m) height (m) Water circulation basin 18 × 4 × 2.1 0.3 (Ifremer, France) OceanWave Basin 35 × 15.5 × 3 0.4 (Plymouth, UK) Flowave TT (UK) ϕ 30 × 2 (circular) 0.7 OTRC (USA) 45.7 × 30.5 × 5.8 0.9 Marin (Netherlands) 45 × 36 × 10.2 0.2 HRWallingford 75 × 8 × 2 1 (flume, UK) MOERI (Korea) 56 × 30 × 4.5 0.8 Oceanide (France) 40 × 16 × 5 0.8 In the case of offshore engineering applications, the wave height will not be limited by the water depth but rather by the model scale and by the simulated sea conditions. A good compromise is achieved by setting a maximum wave height for regular waves to about 0.35 m, which will allow for a comfortable range of model scales. For a wave basin like the COB, the capability for oblique wave generation is an extremely relevant aspect which will be achieved by a wavemaker composed of relatively narrow paddles. Investigations were performed regarding the largest paddle width that allows for a satisfactory oblique wave quality. In this case the most demanding conditions are for shorter waves when they are produced at a large angle from the wavemaker normal direction (Andersen & Frigaard, 2014). The wave quality is typically specified as a spurious wave content for a certain wave period and angle, but to have a clear cut quality criterion and to later verify the accomplishment of these goals is rather difficult. Therefore a maximum paddle width was determined which is in the range of 0.67 m for the case of a snake-type wavemaker and about 0.5 m for a box-type wavemaker, respectively. These values would allow the COB to produce waves with 1 s periods or longer in any oblique direction with respect to the wavemakers with a satisfactory quality. 3.4 Current generator One of the unique characteristics of the COB is the capacity of producing combined waves, current and wind. The literature and internet bibliographic search showed that there are not many combined flow and current facilities. Journal publications regarding experiments with combined waves and currents are also scarce (Lorke et al., 2011; Toffoli et al., 2013). It is important to note that an off-the-shelf solution for the current system does not exist. The design of the system requires a tailor-made solution considering the basin layout and target flow rate. The target current velocity is based on the flow conditions in the Belgian

Table 3. Current systems in existing wave basins.

Dimensions Flow rate Current Name (L ×W × D) (m) (m^3/s) velocity (m/s) Coastal basins DHI (shallow basin, Denmark) 35 × 25 × 0.8 1.2 LNHE Chatou (France) 50 × 30 × 0.8 1.5 NRC Canada 47 × 30 × 0.8 1.6 HRWallingford (UK) 27 × 55 × 0.8 1.2 Pacific basin (Deltares, Netherlands) 22.5 × 30 × 1 1.8 Franzius-Institute (Germany) 25 × 40 × 1 5 0.3 QU Belfast (UK) 16 × 18 × 1.2 0.25 Cedex (Madrid, Spain) 34 × 26 × 1.6 0.2 COB (Ostend, Belgium) 30× 30× 1.4 11.2 0.4 Offshore basins DHI (offshore basin, Denmark) 20 × 30 × 3 0.2 Ocean Engineering Tank (Japan) 50 × 10 × 5 0.2 Marin (Netherlands) 45 × 36 × 10.2 0.5 MarinTek (Norway) 80 × 50 × 10 0.25 CCOB Cantabria (Spain) 30 × 44 × 3 18 OceanWave Basin (Plymouth, UK) 35 × 15.5 × 3 0.3 Flowave TT (UK) ϕ 30 × 2 0.8

coastal waters, characterised by tidal currents with a typical depth-averaged flow velocity of about 1 m/s. Considering a maximum scaling factor of about 1:8 the flow velocity in the model is reduced to 0.4 m/s. The current generation system should produce a steady current with an almost uniform depth-profile in a uni form water depth of up to 1.4 m, requiring a total flow of approximately 11 m^3/s. A screening of existing lab oratory facilities (Table 1) reveals that only few basins are able to generate currents with a velocity exceeding 0.25 m/s in this water depth. However, maximum cur rent speeds depend also on the water depth used for a specific test where a smaller water depth would allow a higher speed.

The information on the quality of the flows that can be obtained by the different current system approaches is scarce. Robinson et al. (2015) presented experimen tal model measurements and numerical simulations for the Edinburgh FloWave TT ocean energy research facility (Davey, 2012), stating that a turbulence level of approximately 10% was achieved. Some velocity profiles were also published for the flow systems of Marin basin in the Netherlands (Buchner et al., 1999; Buchner et al., 2008), Marintek in Norway (Stans berg, 2008; Li, 2014), COAST laboratory in Plymouth University, UK (Collins et al., 2013) and the Ocean Engineering Tank of Japan (Waseda, 2005; Toffoli et al., 2013). The design of the COB current system is aimed at achieving a higher flow quality than that of existing infrastructure.

Current and wave facilities can be divided into three groups: jet induced flows, pump and pipe systems and flow chambers. The first two systems are more com pact but they involve the presence of high velocities in different parts of their systems, resulting in relatively higher power requirements and even current velocity limitations. For the COB we prioritized obtaining the highest flow quality while keeping the lowest operation cost possible. In this context the use of a flow chamber below the level of the wave tank floor, namely a current tank, was chosen. A scheme of the current system is shown in Figure 3. The current is introduced in the basin through a number of guiding grids flush mounted in the basin floor. Each grid can be replaced by a lid when the current system is not used. The approach taken here to obtain a uniform and steady velocity profile in the wave tank has been to pursue the lowest possible velocities in the current tank and having successive velocity increases as the flow is guided to the wave tank (Idelcik, 1966; Chin and Gerrits, 2001; Hamilton, et al., 1996). The last step is the turn of the flow coming from the bottom of the basin to continue horizontally into the test section which was achieved by designing an inlet guiding grid. In the design process, CFD tools were used to optimise integrally the design of the whole current system. Later the design was verified in a physical model (1:10 scale). Velocity measurements were carried out by means of a particle image velocimetry system. Figure 4 shows a typical vertical velocity profile obtained from the small wave flume of the Civil Engineering Department at Ghent University. The figure shows a rather uniform distribution of velocities from the still water level to about 2/3 of the water depth (around 8 cm), where the velocity decreases from 0.36 m/s to about 0.32 m/s. Further below, in water depths down to 12 cm the velocity decreases to zero velocities as expected. Further model tests and CFD simulations will be run to optimise the design and to eventually suggest the final geometry of the inlet grids.

Figure 3. COB schematic including the wavemaker and the

current system (top: plan view, bottom: cross section).

Figure 4. Velocity profile in model current system.

3.5 Wind generator

The purpose of the wind generator is to generate wind loading on models to be tested. At this stage it was decided to have a frontal flow section of 2 m × 2 m with a maximum wind speed of 15 m/s, as this would generate already a significant dynamic pressure load on a floating structure (130 Pa). The overall esti mated power at maximum speed is 70 kW. The design approach is similar to the one of Ohana et al. (2014), and it consists of a short square duct with an array of fans followed by flow conditioning elements. An exploded view of the planned wind generator is shown

in Figure 5. The flow conditioning design was based

on the wind tunnel design guidelines by Mehta and

Bradshaw (1979) and consists of a metallic mesh, a

honeycomb and a final metallic mesh, with appropriate Figure 5. Wind Generator exploded view. separations in between to allow space for flow redistribution. The function of the honeycomb panel is to reduce streamwise vorticity while the meshes generate a distributed pressure drop that helps uniform the flow field. In contrast with the design presented by Ohana et al. (2014) we decided to have one of the meshes before the honeycomb to ensure the proper performance of this last one. The meshes are going to be made of non-rust metallic wire. 3.6 Water transfer system The water transfer system is needed to fill and empty the COB, but also to set up the working water level.The design flow rate is 1300 m 3 /s, which allows to produce a 10 cm change of the water level in approximately 10 minutes and to take the wave basin from a 1.4 m water depth to zero in approximately 1 hour. A water level setting precision of 1 mm has also been specified. There are three water storage facilities planned, the COB itself, the internal water storage, used for rapid water level adjustments, and the external water storage, used to completely empty the COB. A schematic of the water transfer system is shown in Figure 6. The water transfer system will have to deal with significant variations in water heads when transferring water from one container to another. The system also has to control the water level to within 1 mm accuracy. These requirements call for a specialized system with a valve manifold, frequency controlled pumps and a control and safety PLC. 3.7 Instrumentation An important objective of the COB is to provide state-of-the-art testing and be able to interpret the results. The laboratory will have a large inventory of conventional and state-of-the-art instrumentation for measuring the free surface (i.e. capacitive, resistive,

Figure 6. Water transfer system schematic.

Figure 7. Schematic of the planned data acquisition system

in the COB.

ultrasonic wave gauges), velocity (Acoustic Doppler

Velocimeter, Acoustic Doppler Profiler, micropro peller velocimeter), pressure, stress (axial load cells), wind (ultrasonic anenometer, cup anemometer, barom eter, air temperature sensor), and water depth. In addition, a motion capture system and a 3D laser scanner for topographic mapping system are foreseen.

3.8 Data acquisition system

The Data Acquisition System (DAQ) includes the analogue signal input from the instruments, the com munication with the autonomous systems, the display and processing of signals and other information, and the data storage and back-up. The autonomous sys tems are the water transfer system, the wavemaker, the current system, and the wind generator. Figure 7 illustrates the general concept.

The DAQ will have to cope with different experi mental setups. The number of instruments connected to the analogue signal inputs may vary from one test to the next as well as the measurement positions.

We therefore adopted a system with three acquisition "boxes" connected to an acquisition server by Ether net. Each box offers an assortment of signal acquisition modules covering different possible sensors and acqui sition speeds. The boxes can be easily repositioned in the COB to reduce analogue signal cabling to a min imum. There will be a total amount of 160 analogue input channels, 24 counter channels and 32 bridge or resistance channels.

There will be two fixed locations where the oper ation of the DAQ is possible: from the viewing room

Chin, S. N. and Gerrits, J., 2001. Design and Experimental Verification of an Exit Turning Vane for Improved Current Generation in the National Research Council, Institute for Marine Dynamics' Ocean Engineering Basin.

Davey, T., Bryden, I., Ingram, D., Robinson, A., Sinfield, J.,

and Wallace, A., 2012. The All-Waters Test Facility–a new resource for the marine energy sector, 4th international conference on ocean energy, Dublin, Ireland.

Hamilton, D.G., Rosati, J.D., Fowler, J.E. and Smith, J.M., 1996. Design Capacity of a Longshore Current Recir culation System for a Longshore Sediment Transport Laboratory Facility, Coastal Engineering Proceedings, 1, 25.

Idelcik, I. E., 1966. Handbook of hydraulic resistance, ERDA Div. Phys. Res. ss.

Li, Y., Chen, J. and Ren, N., 2014. The Generation of Ocean Current in a Test Basin, Machines Review, 1, 1, 1–6.

Lorke, S., Brüning, A., Van der Meer, J., Schüttrumpf, H., Bornschein, A., Gilli, S., Pohl, R., Spano, M., Rˇîha, J., Werk, S., and Schlütter, F., 2011. On the effect of cur rent on wave run-up and wave overtopping. Coastal

Engineering Proceedings, 1, 32, 13.

Mehta, R.D. and Bradshaw, P., 1979 "Design rules for small

low-speed wind tunnels." Aeronautical Journal 83, no.

827: 443–449. Ohana, J., Le Boulluec, M., Peron, E., Klinghammer, C., Tancra, A., Mansuy, E., 2014 "Open jet blower type wind generator with variable wind speed capability for physical model testing of offshore structures", Coastlab 14, pp. 375–384. Robinson, A., Ingram, D., Bryden, I. and Bruce, T., 2015. The effect of inlet design on the flow within a combined waves and current flumes, test tank and basins., Coastal Engineering, 95, 117–129. Stansberg, C.T., 2008. Current effects on a moored floating platform in a sea state, ASME 2008 27th International Conference on Offshore Mechanics and Arctic Engineering, American Society of Mechanical Engineers. Toffoli,A., Waseda,T., Houtani, H., Kinoshita,T., Collins, K., Proment, D. and Onorato, M., 2013. Excitation of rogue waves in a variable medium:An experimental study on the interaction of water waves and currents. Physical Review E, 87(5), p.051201. Waseda, T., 2005. Experimental investigation and applications of the modulational wave train. In Proceedings of the Workshop on Rogue Waves (pp. 12–15). Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Idealized model study on tidal wave propagation in prismatic and converging basins with tidal flats

Thomas Boelens & Tom De Mulder

Hydraulics Laboratory, Department of Civil Engineering,

Faculty of Engineering and Architecture, Ghent University, Belgium

Henk Schuttelaars

Delft Institute of Applied Mathematics, Delft University of Technology, The Netherlands

George Schramkowski

Flanders Hydraulics Research, Belgium

ABSTRACT

The tidal flow in an estuary, and consequently the mor phodynamics and ecology, is highly dependent on the basin geometry (Ridderinkhof et al. 2014).The charac teristics of tidal wave propagation in prismatic basins are well known, and also the effect of converging basins has been studied to some depth. The influence of tidal flats on tidal wave propagation has been stud ied using 1D models, while in this paper, both a 1D and a 2D model will be applied. The latter approach allows to account for the water motion above the tidal flats, in contrast to a 1D model where the tidal flats are modelled as storage areas.

To gain some fundamental insight, an idealized model (which is fast, compared to complex proces based models and easy to analyse) has been used, that describes the water motion in a semi-enclosed (converging) basin by means of the shallow water equations, forced by prescribed free surface eleva tions at the entrance ($x = 0$). The main focus of this paper is to study the influence of tidal flat geometry on the spatial structure of different tidal harmonics and of tidal asymmetry between ebb and flood peri ods. This work represents a first, validation step in the

development of a 2D idealized model for the iden

tification of morphodynamic equilibria. The Finite

Element Method (FEM), was used to spatially dis

cretize the governing equations, in which the physical

variables are expanded in their tidal constituents. The

rectangular and converging tidal basins, that are con

sidered here, can thus be easily extended to more

general geometries.

After a favorable comparison with other models

in literature, the tidal hydrodynamics was studied for
Figure 1. Amplitude ratio A M 4 A M 2 [−] of the free surface elevation as a function of x/L, the normalised distance along the channel, wtih entrance at x/L = 0, for different normalised widths of tidal flats ▯b 0 /b 0 (b 0 is the width of the main channel at the entrance, ▯b 0 is the width of the tidal flats at the entrance). different values of bottom roughness and for different widths of the tidal flats. As an example, the free surface elevation amplitude of the internally generated overtide M 4 (quarter-diurnal), relative to that of the M 2 (semi-diurnal) component, prescribed at the entrance, is given in Fig. 1, for a converging channel. Though discrepancies exist between the 1D and 2D results, both models show that the amplitude ratio increases towards the closed end of the basin (x = L). REFERENCE Ridderinkhof, W., H. de Swart, M. van der Vegt, N. Alebregtse, & P. Hoekstra (2014). Geometry of tidal inlet systems:A key factor for the net sediment transport in tidal inlets. Journal of Geophysical Research: Oceans 119(10), 6988–7006.
Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Safety standards for the coastal dunes in The Netherlands

R.P. Nicolai, M. Kok, A.W. Minns & S. van Vuren

HKV Consultants, Lelystad, The Netherlands

C.J.J. Eijgenraam

CPB Netherlands Bureau for Economic Policy Analysis, The Hague, The Netherlands

ABSTRACT: In 2014, the Dutch Delta Programme introduced new safety standards for the Dutch primary flood defence system that were derived using a risk-based approach. The Dutch primary flood defence system consists of a network of dykes (US: levees) and coastal dunes. The new safety standards take into account both the probability of inundation through flooding and the consequences of such a flood. The Dutch Delta Programme has derived the standards based on the risks for individual and group mortality (solidarity principle) and on a cost-benefit analysis (i.e. economic optimum). A full cost-benefit analysis is a data-intensive and a labour-intensive method, and therefore a simple, pragmatic approximation formula, referred to as the direct assessment approach, has been applied. The direct assessment approach yields a good estimate for economically optimal safety standards for dykes in the Netherlands. The direct assessment approach has subsequently been applied to the coastal dunes in the Netherlands. However, the correctness and applicability of this approximation to these locations has not been based upon hard empirical or analytical evidence. This paper shows that the direct assessment approach is indeed applicable to coastal dunes in the Netherlands but not as a direct consequence of its applicability to dykes but rather due to the specific nature of sand nourishments, which are used to reinforce the coastal dunes.

1 NEW FLOOD SAFETY STANDARDS

Since 2013, the Dutch Delta Programme has under taken flood risk assessments for all of the Dutch primary flood defences. These assessments resulted in new proposed flood protection standards (DPV, 2014). To obtain economically optimal flood probabil ities, the costs required to reinforce the primary flood defences and the economic consequences of a possible flood have been balanced. This cost-benefit analysis (CBA) is based on a decision model in Eijgenraam (2006), which was implemented in a nation-wide CBA for flood safety (Eijgenraam, 2014; Kind, 2014). The resulting economically optimal flood probabilities can be regarded as the successors of the dyke heights that were derived by the Dutch Delta Committee in 1960 (Van Dantzig, 1956).

A full CBA is a data-intensive and labour-intensive computational undertaking. Moreover, for the Dutch coastal dunes, the implementation of a CBA leads to several practical problems, mainly due to the very high current safety levels of these dunes. In the Dutch Delta Programme (2014) a more pragmatic approach has been applied to derive the flood protection standards for coastal dunes. This approach, referred to as the direct assessment approach, was derived from the out comes of a complete CBA for dykes, but has been subsequently used to determine the flood protection standards for coastal dunes. In a very limited number of cases where a full CBA of the coastal dunes has yielded an actual result, a comparison with the direct assessment approach shows some similarity. However, hard empirical or analytical evidence for the correctness and applicability of the direct assessment approach to coastal dunes was lacking. It is, indeed, desirable to gather such evidence in order to assuage the remaining doubts of policy makers. After a brief review of the current Dutch approach to developing flood protection standards in general, this paper analyses the approximations used in the direct assessment approach and discusses their applicability to coastal dunes. Finally, we describe the conditions under which the direct assessment approach is valid for coastal dunes and identify some possible limitations. 2 ECONOMICALLY OPTIMAL SAFETY STANDARDS FOR FLOOD PROTECTION 2.1 Optimal flood probabilities A description of economically optimal safety standards in the Netherlands first requires some background information about the CBA and the definition of optimal flood probability. In the context of flood protection the concept and use of a CBA

Figure 1. Illustration of the cost-benefit analysis for flood

protection of coastal dunes.The x-axis displays the sand nour

ishment volume (m^3/m). The y-axis shows the costs (euros).

Red line: investment costs. Blue line: NPV of expected flood

damage (economic risk). Black line: total costs.

was first introduced by the Delta Committee in their

study of economically optimal safety standards for

Central Holland following the 1953 flood disaster (Van

Dantzig, 1956). In this study, the economically opti

mal solution was defined as the minimum of the sum

of the costs for reinforcement of the flood protection

system and the expected value of the remaining flood

damages (economic flood risk). Figure 1 illustrates

this principle for the flood risk of coastal dunes. Sand

nourishments may be applied to decrease the flood risk as long as the incremental cost for a subsequent reinforcement is lower than the resulting decrease in flood risk. At the optimum, the sum of the total costs of reinforcement and flood damages is lowest and the associated flood probability, or the prescribed safety standard, is economically optimal.

The solution of the dynamic decision model by Eijgenraam (2006) is an optimal reinforcement strat egy, consisting of the size of reinforcements (e.g. increase in dyke height, volume of sand nourishment) and the timing of subsequent reinforcements. This is based on the premise that protection levels will nat urally decrease in time through changes in the water system (e.g., climate change, soil subsidence), ageing of the infrastructure and other factors. The decision model distinguishes between the resulting protection level just after a reinforcement has been applied and the protection level just before the next reinforce ment is required. From an economic point of view these two protection levels can be regarded the opti mal design level (after reinforcement) and the rejection level (before reinforcement). Figure 2 shows these two levels. Pmin denotes the rejection level, which corresponds to the flood probabilities just before the reinforcement moments. Pplus denotes the design level, which corresponds to the flood probabilities just

after the reinforcement moments.

In Eijgenraam (2009) it is argued that the opti

mal yearly middle flood probability is a good starting

point for prescribing the statutory test standard (for

testing whether the actual safety of a dyke ring is Figure 2. Flood probabilities as a function of time. The x-axis displays the date. The y-axis displays the flood probability.The solid line shows the development of the actual flood probability (P) in course of time, both before as well as after reinforcements of the flood protection system.The dotted and dash-dotted lines denote the design level (Pplus) and rejection level (Pmin), respectively. Finally, the dashed line (Pmidden) is the middle flood probability defined by Eijgenraam (2009). Adapted from Duits et al. (2011; Figure 3). above a prescribed minimum safety standard). The main reasons are: – The middle probability only depends on the average cost of the optimal reinforcement, being the cost of dyke heightening per centimetre or the cost of sand nourishment per cubic meter sand per meter dune. Hence, the middle probability does not depend on the breakdown in fixed and variable reinforcement costs. (The fixed costs are the part of the reinforcement costs that do not depend on the size of a reinforcement. Variable costs increase with the size of the reinforcement. High fixed costs make large dyke reinforcements economically attractive.) – Exceedance of the middle probability still gives enough time to plan and execute large reinforcement projects before the rejection probability (Pmin) is exceeded. The Dutch Delta Programme (2014) considers the middle probability P midden t (per year) as the economically optimal safety standard. The middle probability can be derived from a CBA and is defined as: in which V t is the cost of damages (euros) due to a flood at time t; and S midden t is defined as the logarithmic average of the expected value of flood damages just before and just after the next reinforcement (euros/year). The expected value of flood damages, also referred to as flood risk, is defined as flood probability x flood damages. 2.2 Direct assessment approach The solutions to the dynamic decision model are the optimal sizes and timing of reinforcements. With these

outcomes the associated middle probabilities can be calculated. The model requires a large amount of input data, including the economic damages for sev eral flood scenarios, the reinforcement costs and the flood probability as a function of time. The computing time of the solution method increases with the number of different segments of the dyke ring under consid eration. The total length of the Dutch primary flood protection infrastructure is about 3,500 km. Therefore, a practical application of the model requires a faster, pragmatic solution method.

In Eijgenraam (2009), it has been shown that for a single dyke section S midden t is a linear function of the discount rate and the average reinforcement costs of the optimal dyke heightening u:

in which δ is the annual discount rate (1/year); $1/\theta$ is the size of a reinforcement action (meters dyke height ening) that reduces the expected flood damages by a factor of $e \approx 2.72$; and I (u) denotes the costs (euros) of the optimal reinforcement action u (meters dyke heightening).

Now, let us suppose the optimal reinforcement action reduces the flood probability (and thus the expected flood damages) by roughly a factor of 10 and the average reinforcement costs I (u)/u (euros per meter dyke heightening) are almost constant close to the optimum. Under these assumptions, the middle probability in Eq. (1) can be approximated by

where u 10 is the reinforcement action that reduces the flood probability by a factor of 10. Eq. (3) can also be written as:

in which C is a constant; and for which $C = \delta/\ln(10) \approx 1/42$ for an annual discount rate of 5.5% (the gov ernment prescribed discount rate used in cost-benefit analyses).

In Kind (2014) a comparison was made between the middle probabilities derived from the full CBA for flood protection by dykes and middle probabilities calculated from Eq. (4). A good match was found with a value of the constant C close to 1/38. It is noted that this value is only 10% larger than the value of 1/42 given above.

Eq. (4) with C = 1/38 is referred to as the direct assessment approach. It only requires two inputs to approximate the middle probability: (i) the costs of a reinforcement action that reduces the flood probability of a water defence section by a factor of 10 and (ii) the economic flood damages in the year 2050 (Dutch Delta Programme, 2014). 3 SAFETY STANDARDS FOR COASTAL DUNES 3.1 Why is the direct flood assessment approach used for coastal dunes? In general, the coastal dunes in the Netherlands are very large and robust, and the flood probability for areas protected by these dunes is very low.

In fact, the current level of protection provided by the coastal dunes is generally much higher than the statutory requirements (Vuik et al., 2015). Unfortunately, the very low flood probabilities in areas protected by coastal dunes complicate the CBA. In this case, the cost calculations can only be done for a small set of flood probabilities. This set is then too small to derive a reliable estimate of the middle probability. Some example calculations of a full CBA for the coastal dunes at a number of locations along the Wadden Sea and the North Sea only yielded an actual result in a very limited number of cases (Van Vuren et al., 2013). The direct assessment approach is computationally less intensive and always yields results at all locations. It was noted that in the cases that a full cost-benefit analysis yielded an actual result, this result was in fact quite similar to the result from the direct assessment approach. The direct assessment approach has subsequently been used as the only method for the derivation of safety standards for coastal dunes. Although the direct assessment approach has been demonstrated to be widely applicable to dykes, there was no hard empirical or analytical evidence to support its correctness and applicability to coastal dunes. Obviously, sand nourishment and dyke heightening are quite different reinforcement measures. Also, the failure mechanisms of dykes are significantly different to the failure of a coastal dune, which occurs typically through erosion. Differences can also be expected between the cost functions of reinforcement measures as well as the relation between the reinforcement action (sand nourishment volume) and the flood probability. In particular, a question can be raised about whether a sand nourishment that reduces the flood probability by a factor of 10, as assumed in Eq. (4), is in fact near the optimum reinforcement strategy. 3.2 Optimal reinforcement of the dunes The direct assessment approach is valid in situations where a dyke ring or part of a dyke ring consists of a single section (Eijgenraam, 2009). In DPV (2014), specific sections have been defined for the derivation of safety standards that fulfil this validity requirement. Furthermore, the direct assessment approach assumes that: 1. the optimal reinforcement is an action that reduces the expected flood loss by a factor of 10, 2. close to the optimal reinforcement action, the average reinforcement costs (e.g. euros per meter dyke heightening) are approximately constant, and,

Figure 3. Illustration of the relation between coastal dune

probability of failure and sand nourishment volume. On the

x-axis the sand nourishment volume (m^3/m) and on the y-axis

the probability of failure (1/year) is displayed.

3. the probability of failure is log-linear in the rein

forcement actions.

If the average reinforcement costs are nearly con

stant on the entire domain, then Eq. (3) is in fact nearly

exact and the size of the optimal reinforcement action

is not relevant for determining the middle probabil

ity. As a consequence, the assumptions (1) and (2)

can be replaced by a stronger assumption: the average

reinforcement costs are nearly constant.

We shall now argue that in the case of coastal

dunes the average reinforcement costs are almost con

stant and that the log-linear relation exists. For coastal

dunes, the reinforcement is in the form of sand nour

ishment, which is measured as a volume of sand per

meter coastline (m^3/m) that is placed on the beach

and foreshore in front of coastal dunes. With respect

to the associated reinforcement costs, the fixed costs

consist mainly of initiation and preparation costs and

are quite low compared to the variable costs (Van

Vuren et al., 2013). The variable costs only depend

on the required sand nourishment volume per meter

of coastline, because the cubic meter price of sand

does not depend on the total volume of sand in the reinforcement. Therefore, the average reinforcement costs (euros/m 3 sand) are constant in any reasonable reinforcement.

In Giardino et al. (2014) it is shown that in the Netherlands the coastal dune probability of failure is approximately a log-linear function of the sand nourishment volume. This is illustrated in Figure 3. It then follows that the direct assessment approach is applicable to the Dutch coastal dunes. It should be noted that the reinforcement action for coastal dunes has been defined as a volume of sand per meter of coastline. The total reinforcement costs must therefore be adjusted to include the length of coastline under consideration. Therefore, Eq. (2) becomes (see also

Eijgenraam, 2015): where 1/θ is a sand nourishment (m 3 /m) that reduces the expected flood damages by a factor of e ≈ 2.72, u is a sand nourishment action (m 3 /m of coastline), I d (u) are the associated costs (euros/m of coastline) and l is the length of the coastline under consideration (meters). The middle probability for dunes is thus computed as Summarizing, the direct assessment method with the costs of a reinforcement action u 10 and C = 1/38 is applicable to the Dutch coastal dunes. The interesting part of the proof is that the size of the optimal reinforcement action does not need to be close to u 10 , provided that the average costs are nearly constant for any reasonable reinforcement action.The optimal sand nourishment volume is most likely smaller than u 10 due to the comparatively small fixed costs of such a reinforcement. 3.3 Results In 2014, the direct assessment approach (with C = 1/38) has been applied to derive economically optimal safety standards for the Dutch coastal dunes. The values for the North Holland and South Holland coast vary between 1/1,000 per year and 1/30,000 per year. For some coastal dune sections along the coast of Zeeland and the Wadden Sea islands, the values are higher because the expected flood damages are smaller. The current levels of protection against flooding that are provided by the existing coastal dunes are almost always much higher (i.e., lower flood probabilities) than these proposed safety standards (by at least a factor of 10). 4 CONCLUSIONS This paper reflects upon the reliability and applicability of the direct assessment approach for deriving economically optimal safety standards. Following the Dutch Delta Programme we consider the middle probability derived from a societal CBA as the economically optimal safety standard. Due to a number of problems with the application of a full CBA to coastal dunes, the direct assessment approach has subsequently been used to derive the middle probabilities in these locations. It has been shown that although the direct assessment approach was only derived for the reinforcement of dykes, it also provides appropriate results for coastal dunes. The governing assumptions of the direct assessment approach applied to dyke reinforcement require: 1. the water defence under consideration consists of a single section,

2. the optimal size of a reinforcement action is an action that reduces the flood probability of the section by about a factor of 10,

3. the average reinforcement costs are almost constant near the optimum, and

4. the failure probability of dykes is log-linear in the dyke heightening.

We have demonstrated that after adapting the above assumptions to the case of coastal dunes the direct assessment approach is indeed applicable to coastal dunes, because:

1. the water defence under consideration consists of a uniform length of coast,

2. the average sand nourishment costs are approximately constant,

3. the failure probability of coastal dunes is log-linear in sand nourishment actions.

These results justify the application of Eq. (6) to coastal dunes (with C = 1/38). It also follows that the size of the optimal sand nourishment action is not rele vant for deriving the economically optimal flood safety standard. The method does not give any insights into the optimal sand nourishment volume. The optimal sand nourishment volume is most likely small, because the fixed costs are low compared to the variable costs, which depend only on the required sand nourishment volume per meter of coastline. Hence, frequent sand nourishment actions seem to be economically optimal. This corresponds with the present coastal maintenance practice of small-scale sand nourishment actions for the Dutch coast.

ACKNOWLEDGEMENTS

The authors would like to thank IlkaTánczos (affiliated with Rijkswaterstaat, the executive body of the Dutch Ministry of Infrastructure and the Environment) for useful comments on a previous draft of this paper.

van Dantzig, D. (1956). Economic decision problems for flood prevention. Econometrica, 24, 276–287.

Dutch Delta Programme (2014). Werk aan de Delta. Beslissingen om Nederland veilig en leefbaar te houden.

Deltaprogramma. Ministry of Infrastructure and the Environment, and Ministry of Economic Affairs (in Dutch). http://deltacommissaris.nl/Images/Deltaprogramma%20 2015%20Nederlands_tcm309-358049.pdf [Accessed April 2015]. DPV (2014). Synthesedocument Veiligheid. Achtergronddocument B1. Deltaprogramma, deelprogramma Veiligheid. Ministry of Infrastructure and the Environment and Ministry of Economic Affairs (in Dutch). http://www. deltacommissaris.nl/Images/DP2015%20B1%20Synthese document%20Veiligheid_tcm309-358052.pdf [Accessed April 2015]. Duits, M., Kind J.M., and Bak C. (2011). Maatschappelijke kosten-batenanalyse Waterveiligheid 21e eeuw. Bijlage C: Wiskundige achtergrond MKBA model. Technical report (in Dutch). Deltares. http://www.deltaportaal.nl/ programfiles/4/programfiles/MKBA_WV21_Bijlage_C. pdf [Accessed April 2015]. Eijgenraam, C.J.J. (2006). Optimal safety standards for dikering areas. CPB discussion paper No. 62. http://www.

Impact of the sea level rise on low lying areas of coastal zone: The case of Batu Pahat

A.B. Yannie, A.H. Radzi & A. Dunstan

Coastal Management & Oceanography Research Centre, National Hydraulic Research Institute of Malaysia (NAHRIM), Ministry of Natural Resources & Environment (NRE), Selangor, Malaysia

W.H.M. Wan Mohtar

Department of Civil And Structural Engineering, Faculty Of Engineering, University Kebangsaan Malaysia (UKM), Selangor, Malaysia

ABSTRACT: This study investigates the impact of sea level rise at the coastal zone area of Batu Pahat, Johor which identified as low lying area, located at the south west of Peninsular Malaysia. The coastline of Batu Pahat has experiencing severe erosion and significant land loss due to being inundated by sea water. The inundation profile was predicted using Numerical modelling developed by Danish Hydraulic Institute (DHI) familiarly

known as MIKE 21 Flexible Mesh – HD. The model predict the inundation profile for the year of 2020 and

2040 along the Batu Pahat coastline, where the hydrodynamic condition in 2013 was referred as baseline year.

The model boundary was design meeting the interest or critical area of the coastal segment of Batu Pahat with

dimensions of 42.7 × 126 km 2 . Tidal, current and wave data were calibrated and the error ranges are within

the acceptable limit. From the model simulation, current speed has increase from 0.2–0.39 m/s (baseline) to

0.36–0.9 m/s in year 2020 and has reached up to 1.8 m/s by the year 2040. On the other hand, inundation map

produced showing 1.08% of the developed areas including industrial zone will be affected by the year 2020.

The percentage for population area, road networks and mangroves forest is approximately 2.04%, 18.18% and

29.08% respectively for the year 2020. Hitting the year 2040 however, the numbers are expected to increase up

to 2.47% for population, 21.21% for road network and about 33.3% for mangrove forest areas.

Keywords: sea level rise, Batu Pahat, hydrodynamic model, inundation risk map

1 INTRODUCTION

Global warming, climate change and sea level rise

(SLR) is indeed currently taking place and showing

it cumulative impacts. The SLR event is constantly

being discussed and researched globally. SLR depends

on various factors and rely heavily on climate change

acceleration rate.Among the major contributor leading

towards accelerating SLR progression are tempera ture, atmospheric pressure, cyclonic cycles and also ocean circulation.

It was estimated that approximately 400 million people who live within 20 m of sea level and within 20 km from a coast along the shoreline or below of 10 m of Mean Sea Level (MSL) will be at the risk of experiencing the impact of the SLR (Small et al., 2000; NPR, 2007; Anthoff et al., 2006; Rowley et al., 2007). It is also expected that at the end of the century, continent of Asia facing significant adverse impacts on its coastal ecosystems, particularly at the low lying areas (Simon et al., 2000, IPCC 1996a, IPCC 1996b and Cruz et al., 2007). The event will not only lead towards a series of inundation risk potential but also worsening beach erosion, more frequent flooding episodes and resulting coastal land area losses particularly the small and low-lying islands (IPCC 1996a, IPCC 1996b and Cruz et al., 2007).Apart from the imminent consequences, coastal land use activities within the coastal zone such as coastal forest (mangroves), agricultural, aquaculture, urbanization and road networks will also be affected. This paper presents the findings of inundation risk area potential due to local SLR impacts from baseline (2013) towards the year of 2020 and 2040. The impact assessment mainly focuses on road networks, population, mangroves forest and development area within the low lying area of coastal segment at Batu Pahat districts, which located in the Malacca Strait. The inundation risk maps will supply necessary and vital information on coastal development planning for the relevant agencies such as Town and Country Planning Department Public Works Department, Department of Irrigation and Drainage, Hydrographic Centre and also the Non-Governmental Organisations (NGOs) in their development plan at the coastal areas.

The main objectives of the study are

1.1 The study area

Batu Pahat district of Johor State located at southern region of Malacca Straits and experiencing semidiur nal tide. The Malacca Straits is much important straits especially for its international shipping route linking that connecting the Indian Ocean(via the Andaman Sea) to the South China Sea and the Pacific Ocean (Chia, 1998; Beckman, 2001; Paw and Diamante, 1995).

Other than its navigational purposes, the West Coast of Peninsular Malaysia is dominated by coastal plains and basins formed by alluvial deposits while only few areas along the coast of Peninsular Malaysia are sandy beaches (EPU, 1985).

As it have a tropical climate; it was strongly influ enced by the Northeast Monsoon that brings rain from December to February, and by a dry Southwest Mon soon from June to August. During the two – inter monsoon period, the weather become unpredictable, while it was influence by semi-diurnal with a tidal range of 1.6 ± 3.7 m depending on location (EPU, 1985).

1.2 Trends of the sea level rise

The trend of sea level rise has shown that the mean global sea level is increasing in the range of 1 mm/yr to 2.5 mm/yr for the past 150 years. However, stud ies by Gornitz, (1995a) and Warrick et al., (1996) have estimated that global SLR rate should be around 1.8 mm/yr, taking into consideration the global climate warming of 0.5 ∘ C as reported by the IPCC1996a).

In Malaysia, the study of SLR was conducted by National Hydraulic Research Institute of Malaysia (NAHRIM) in 2010. The study shows, SLR rate within the territorial waters of Malaysia ranging from 2.73 mm/yr to 7.00 mm/yr. The rate of SLR in Malaysia waters were obtained using the linear trend analysis on historical datasets of satellite altimetry data (1993–2010) at 30 identified spots within the Malaysia waters (NAHRIM, 2010).

The projected SLR rate was found compounding higher at the North West and North East regions in Peninsular Malaysia, whereas East Coast of Sabah waters was recorded for the highest value. Station at Southern of Johore show the lowest of SLR recorded with 0.066 m for Malaysia waters in the year of 2040. Figure 1 shows the projected of the SLR rate for 2040 in Malaysia waters.

1.3 Coastal erosion activity

The initiative to assess coastal morphological char acteristics and erosion profile across the coastal area of Malaysia was initiated in 1984 and the report

was named National Coastal Erosion studies (NCES),

which completed in 1985. The study summarized about 29% or 1,380 km of 4,809 km of the country’s coastline was already facing erosion problems. In another study done by Mohamad et al.,(2014), 3.3% of the shoreline in the Peninsular Malaysia was classified as being extremely vulnerable towards erosion. As the coast of Batu Pahat predominantly muddy coast, a major socio economic activity is agricultural crops of coconut and fruits farms.This study also show that muddy coast of the area can reach up to 1.5 km of muddy flat towards the sea, which is not surprisingly have prepared a suitable place for the mangrove ecosystem. However, some part of the area has facing the erosion, which expose the mangrove in the shore area to the vulnerability state. 1.4 Coastal land loss The factors that leads to erosion are vary but definitely few factors such as ocean waves, currents, and tides will reshape the local shorelines. Continuous erosion and shore degradation exacerbated by SLR and storm surges with highest storm surges is indeed rapid changes of the shorelines. This acceleration of the land loss in the coastal areas can be 100 times greater or more than the rise of the sea level itself (Cooper et al., 2008; Zhang et al., 2004). In isolated cases, due to high amplitude of the storm surges, catastrophic damages to houses and critical infrastructures in the near coast area were often the main caused (UCS, 2013). Batu Pahat is definitely experiencing and witnessed SLR along the coastal stretches. This can be seen from the coastal morphologies changes within the last 15 years. Information from the census conducted among the coastal residences have indicated irrevocable losses of land part wherein some coastal stretches several hectares of land has now become part of the sea. 2 METHODOLOGY AND DATA Generally, the method in this study is design to meet the ultimate goal of this study enlisted below: • To conduct primary marine data collections for model input and calibration at Batu Pahat coastal regime; • To predict future SLR impacts towards the coast of Batu Pahat district; • To assess physical vulnerability in the presence of SLR phenomena across Batu Pahat coastal regime and • To produced Inundation Risk Map for Baseline (year 2013), 2020 and 2040. 2.1 Management Unit for the risk estimation In order to carry out a detailed risk estimation of the study area, it is necessary to divide the study area into Management Units (MU) prior estimating the risk impact on the study area. The coastline of Batu Pahat is divided into six of Sub Reaches (SR): SR1 to SR6,

Figure 1. Projected sea level rise along Malaysia coastal area by year 2040 and the study area shown in red rectangle box.

Figure 2. Management Unit used along the study shoreline.

which then sub divided into 17 Management Units

(MUs): MU1-1 to MU6-3 (Figure 2).

The division of the Management Units (MU) is

based on the existing coastal features and land use.

This is done by analyzing the four factors of road net

works, population, mangrove forest and development

area for each MU's and then the percentage of the risk

are calculated as below:

2.2 Model setup

This study has using MIKE 21 Flow Model FM as

a primary tool. The model was used to predict the

changes of coastal current flow and near shore wave

with successive increases in sea levels. Figure 3. Setup of the modal boundary. Three boundaries for the water level were created, Northern, Western and Southern to design the water entry point inside the model. The model area size is 42.7 km × 126 km as depicted in Figure 3. The bathymetry measurement was extended from 0 m to more than −63 m below MSL (see Figure 4). Smaller range can be seen within the area of interest at Sg. Batu Pahat to Sg. Tampok with the average bathymetry profile was not more than −12 m using extracted data from the C-map. There are two sets of the tide gauge(TG1 and TG2) has been installed in the coastal area which has attached to the ADCP. Each of an installation is in the depth of 10.5 m to 12.5 m from MSL, for better measured data of the tidal measurement, current speed and its direction. In this study, simple and liner statistical has been used to analyzing the data to get the idea of measured and simulation data pattern.

3 RESULTS AND DISCUSSIONS

3.1 Model calibration and verification

The baseline model was simulated during the South west monsoon event by incorporating the wind speed of 7 m/s and wind direction from 300 ∘ , respectively. In order to verify the accuracy of the model, the model results were statistically analyses compared to observed water level, current speed and current direction.

Simulated model has shown that the Simulation of the HD has been improved with the Spectral Wave (SW) module. Tidal, current speed and current direc tion of the HD model were calibrated and the error ranges are within an acceptable limit (see Figure 5). These acceptable limits are referred to the Guideline on the hydrodynamic modeling preparation by JPS (2013) which are 10% of error calibration on the tidal are allowable, 20% on current speed and 20 degree of error on the current direction data.

3.2 Analysis and inundation risk map

Analysis of the simulation results showed the cur rent speed increased from baseline ranging from

Figure 4. Local bathymetry conditions at the Batu Pahat coastal area.

Figure 5. Results of the calibration process. 0.2–0.39 m/s to 0.36–0.9 m/s in 2020 and up to 1.8 m/s in the year 2040.

Based on the results of simulated hydrodynamic model for the baseline condition, the rise in the sea level could lead to saline water flooding and intrusion during dry season extended up to 4.5km inland. In more serious case, the saline water flooding and its intrusion can extend up to 12.9km inland at the canal of Sg. Simpang Kanan. The most vulnerable coastal segment located at Minyak Beku Ecotourism Jetty, Sg. Ayam village Teluk Segenting, Sg. Koris and Sg. Punggor. The reason is because most of these areas are either have less coastal revetment structures or the existing structure were damaged or design failure which leads to insufficient height to prevent salt water intrusion and overtopping. Concrete evidences of this situation can be seen in Photo 1 to 4 during the site visit held on 8 November 2014 at the bottom this page. Note that at this time, continuous heavily rain has occurred and coupled with the highest spring water level (King tide) and storm surges. Analysis on the morphological data has identified three (3) locations with good resilience towards erosion due to effective structural design and good Photo 1. Overflow of the sea in the Teluk Serting during normal high tide. Photo 2. Saltwater has inundated the Kg. Sg. Ayam on 8 November 2014.

engineering practice. This situation can be observed

at places such as Perigi Batu Pahat jetty (at MU2-2);

Sialu Island and Tanjung Segenting (at MU2-3); and

Tanjung Laboh and Tikus Kecil Island (at MU4-2).

Observation and data analysis have indicate that

three MUs has classify in flat beach profile, i.e. MU2

3, MU4-1 and MU4-2, while fourteen MUs out of 17

are classify in a gently slope beach. For aMU2-3 and

MU4-3 does not require any proposed adaptation mea

sures as it is because the existing structure is relatively

high and the potential impact of sea level rise over this

Photo 3. Salt water intrusion into coconut plantations have

led to the death of plants.

Photo 4. The arrow indicates the distance to an abundance

of sea water into recreational area at Sg. Punggor.

Table 1. Summary of risk potential at Batu Pahat coastal land use area. Impact of SLR Inundation area Development Population Road networks Mangroves

Study Area (km 2) (km 2) (person) (no.) (ha)

Total Area or No. involved 468 53.05 38,750 33 12,056.02

Baseline (2013) 10.81 0.08 50 3 2,094.71 (2.31%) (0.15%) (0.13%) (9.09%) (17.37%)

0.028m SLR (2020) 27.23 1.08 920 6 3,506.47 (5.82%) (2.04%) (2.37%) (18.18%) (29.08%)

0.066m SLR (2040) 30.80 1.40 1,300 7 4,015.72 (6.58%) (2.64%) (3.35%) (21.21%) (33.30%) area is less significant. In addition, the rocky beach area are able to avoid the ravages of erosion. In other hand, MU3-1 and MU3-2 are likely to require adaptation measures proposed, more drastic transfer of the existing population of the area at risk to higher ground and to secure area. All data and detailed impact analysis of the SR and MU for coastal Batu Pahat are summarized in Table 1. About 0.72% of the developed areas including industrial region will be affected by the sea level rise in year 2020.The percentage of inundated area of population, road network and mangroves are 0.56%, 18.18% Figure 6. Potential of the development area will be inundate by year 2040. Figure 7. Potential of the road networks will be inundate by year 2040.

Figure 8. Inundation potential on the mangroves area by

year 2040.

Figure 9. Inundation potential of population involved by the

year 2040.

and 29.08%, respectively by the year 2020. Hitting the

year 2040 however, the numbers are expected to be

increased to 2.47% for population, 21.21% for road

network and about 33.3% of mangrove areas will be diminished.

Figure 6 to 9 show the Inundation Risk Map for the baseline (year 2013), projected for year of2020 and 2040 which show the sea level are 0.028m and 0.066m, respectively.As the study are to produce inun dation map until year of 2040, for the 0.066m of SLR show that 6.58% of the area are inundated in that year, while up to 2.64%, 3.35%, 21.21% and 33.30% of the development area, population, road networks and mangroves are affected, respectively.

4 CONCLUSION

The study indicates tidal, current and wave data cal ibrated has show the error ranges are within the acceptable limit and in good agreement. HD model result, in which made into the current speed map, indicate that the current speed has increase from 0.2-0.39 m/s (baseline) and has reached up to 1.8 m/s by

NAHRIM (2010). The study of the impact of climate change on sea level rise in Malaysia (Final Report). Seri Kem bangan, Selangor: National Hydraulic Research Institute Malaysia (NAHRIM).

NPR (2007). Environment. Study: 634 Million People at Risk from Rising Seas., March 28, 2007: Nell Greenfieldboyce.

Rowley, R.J., Kostlenick, J.C., Braaten, D., Li, X. and Meisel,

J. (2007). Risk of rising sea level to population and land

area. EOS Transactions, 88, 105-107. Small, C., Gornitz, V., Cohen, J.E. (2000). Coastal hazards and the global distribution of human population. Environ. Geosci. 7, 3-12. UCS (2013). Causes of Sea Level Rise. Fact Sheet.April 2013. Union of Concerned Scientists (UCS), Cambridge, MA: www.ucsusa.org/sea levelrisescience. Luqman Haziq (2010). Kebun kelapa 'ditelan' laut.Johor: Utusan Online.
Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

A database of validation cases for tsunami numerical modelling

D. Violeau, R. Ata, M. Benoit [1] & A. Joly

EDF and Saint-Venant Hydraulics Laboratory, Chatou, France

[1] Now at IRPHE and Ecole Centrale Marseille, Marseille, France

S. Abadie, L. Clous, M. Martin Medina & D. Morichon

Université de Pau et des Pays de l'Adour, Anglet, France

J. Chicheportiche & M. Le Gal

Ecole des Ponts ParisTech, Marne-la-Vallée, France

A. Gailler, H. Hébert & D. Imbert

CEA, DIF, DAM, Arpajon, France

M. Kazolea & M. Ricchiuto

Team CARDAMOM, Inria Bordeaux sud-Ouest, Bordeaux, France

S. Le Roy, R. Pedreros & M. Rousseau

BRGM, Orléans, France

K. Pons, R. Marcer & C. Journeau

Principia, La Ciotat, France

R. Silva Jacinto

Ifremer, Géosciences Marines, Centre de Brest, France

ABSTRACT: This work has been performed by a French national consortium within the framework of the

national project Tandem, with aim to improve knowledge about tsunami risk on the French coasts. Work-package

#1 of this project was the opportunity to build a database of benchmark cases to assess the capabilities of 18

codes, solving various set of equations with different numerical methods. 14 test cases were defined from the

existing literature with validation data from reference simulations, theoretical solutions or lab experiments. They

cover the main stages of tsunami life: 1) generation, 2) propagation, 3) run-up and submersion, and 4) impact.

For each case several of the numerical codes were compared in order to identify the forces and weaknesses of

the models, to quantify the errors that these models may induce, to compare the various modelling methods, and

to provide users with recommendations for practical studies. In this paper, 3 representative cases are selected

and presented with an analysis of the results.

1 INTRODUCTION

This work has been performed by a French national

consortium within the framework of the national

project Tandem (2014–2017), with aim to improve

knowledge about tsunami risk on the French coasts.

The first of the four work-packages of the Tandem

project (WP1 – Qualification and validation of numer

ical codes) was the opportunity to build a database of

benchmark cases to assess the capabilities of various

(industrial or academic) numerical codes.

18 codes were used, solving various set of equa

tions with different numerical methods. 14 test cases

were defined from the existing literature with val

idation data from reference simulations, theoretical solutions or lab experiments. They cover the main stages of tsunami life: 1) generation (from seism or landslide), 2) propagation, 3) run-up and submersion, and 4) impact. For each case several of the numerical codes were compared in order to identify the forces and weaknesses of the models, to quantify the errors that these models may induce, to compare the various modelling methods, and to provide users with recommendations for practical studies. Here, 3 representative cases are selected and detailed with an analysis of the results: – Case GG07: “Russel’s wave generator”, – Case P02: “Solitary wave reflecting on a 2D vertical reef”,

Figure 1. Russel’s wave generator: sketch of the experimen

tal apparatus.

– Case RS04: “Seaside experiment: impact on a urban

area”.

These cases have been chosen as being representa

tive of the main phases of tsunami life, while being

simulated by a significant number of codes from the

project’s list. The participants to Tandems’s WP1 are

the French institutes listed below:

– CEA (Commissariat à l’Energie Atomique), public

research institute, Project coordinator,

– EDF (Electricité de France), private company, WP1

coordinator,

– UPPA (Université de Pau et des Pays de l’Adour),

academic institution,

– ENPC (Ecole des Ponts ParisTech), academic insti tution,

- Inria, public research institute,

- BRGM (Bureau de Recherches Géologiques et Minières), public research institute,

- Principia (French), private engineering company,

- Ifremer (Institut Français pour la Recherche et l'Exploitation de la Mer), public research institute.

It is noted that on the P02 test selected here the waves are propagated on rather short distances; cases with long distance propagation of solitary-type waves or initial disturbances are also included in the test bed.

2 RUSSEL'S WAVE GENERATOR

2.1 Case description

This case is the generation of a solitary wave induced by the vertical fall of a rectangular rigid body and its interaction with the underlying water body. It is based on the experiment published in Monaghan and Kos (1999). This benchmark allows checking the accuracy of a model in a case of strong interactions of a rigid body with free surface. It involves interface reconnec tions and vortices generation and thus requires the use of models able to deal with these kind of phenomena.

In terms of engineering relevance, this case is close to the physics involved in massive cliffs or ice bodies fall into water.

Fig. 1 represents a sketch of the experiment per

formed by Monaghan and Kos (1999). The experiment

was carried out in a 9m long 2D flume, with water

depth D. A 38.2 kg rectangular block (0.4 m tall, 0.3 m

long and 0.39 m wide) is placed just above still water

level at initial time, and then released. Experiments
Figure 2. Russel's wave generator: definitions of B and H (picture from Monaghan and Kos, 1999). were repeated for D = 0.288 m, 0.210 m, and 0.116 m; in each case, the block vertical position and free surface elevation were measured as a function of time at a wave gage located 1.2 m from the leftward extremity of the flume. Values of H and B (Fig. 2) were also estimated from video measurements. To closely reproduce these experiments, in which the block was forced to have a vertical motion, horizontal block velocities must be set to zero in the model. The block has also to be slightly shifted rightward (by 20 mm) as in the experimental set-up, and initially positioned 5mm below the free surface. 2.2 Benchmarking results Three Navier-Stokes models are tested here: – the code THETIS (developed by laboratoire Trèfle, Bordeaux), based on Finite Volume (FV) with the Volume Of Fluid (VOF) technique for free surface tracking, – the code EOLE, developed by Pincipia, based on FV and VOF as well, – the code Sphynx, developed by EDF with help from ENPC, based on the Smoothed Particle Hydrodynamics (SPH) method. To solve the fluid solid interaction, EOLE and Sphynx calculate the resulting pressure force and deduce the motion from Newton's law, while THETIS uses a penalization of viscosity to model the interaction at once. The following discretizations were used in these simulations (L = length of the rigid body): – THETIS_1: $\delta x/L = 0.02$, – THETIS_2: $\delta x/L = 0.01$, – THETIS_3: $\delta x/L = 0.005$, – EOLE_1: $\delta x/L = 0.01$, – EOLE_2: $\delta x/L = 0.005$, – Sphynx: $\delta x/L = 0.016$, where δx is the mesh (FV) or particle (SPH) size.

Figure 3. Russel's wave generator: snapshots of simulations

with THETIS, EOLE and Sphynx (from top to bottom).

Figure 4. Russel's wave generator: solid velocity versus

solid vertical position. Initial water depth = 0.21 m. Sym bols: experiments from Monaghan and Kos (1999); lines: simulations. Times goes on from right to left.

Fig. 3 presents snapshots of simulations carried out with the different models. The key point is the accu racy with which the solid motion is simulated. Fig. 4 presents a comparison of the models on this aspect (the curve is to be read from right to left with increasing time). BothVOF models give satisfactory results while the SPH model slightly underestimates the velocity at the end of the motion. The latter model also shows oscillations at the end, due to the block interaction with the tank bottom.

Table 1 summarizes the wave height simulated by the models a few solid lengths away from the source position, with various initial water depths. Relative

errors are also presented. Experimental values are Table 1. Russel's wave generator: wave height (m) calculated versus measured at x = 1.2 m from the left end of the channel for various water depths. D (m) 0.116 0.210 0.288 Experiment 0.109 0.092 0.093 THETIS_1 0.0966 Error (%) 5 THETIS_2 0.0967 0.1092 0.1024 Error (%) 11 19 10 THETIS_3 0.100 Error (%) 10 EOLE_1 0.085 0.094 0.098 Error (%) 22 2 5 EOLE_2 0.093 0.094 0.101 Error (%) 15 2 9 Sphynx 0.094 0.092 0.092 Error (%) 14 0 1 Table 2. Values of H and B for an initial depth of 0.210 m (see Fig. 2). H (m) B (m) Experiment 0.333 ± 0.01 0.303 ± 0.02 THETIS_1 0.295 0.264 Error (%) 11 13 THETIS_2 0.294 0.270 Error (%) 12 11 EOLE_1 0.317 0.291 Error (%) 5 4 EOLE_2 0.317 0.296 Error (%) 5 2 Sphynx 0.295 0.280 Error (%) 11 8 given with a relatively coarse resolution of 0.01m which prevent from an accurate estimation of the numerical error. Model errors obtained with comparable mesh resolution are presented in bold in the table. The numerical estimations generally show a very good accuracy (within the experimental error bar) except for one simulation per model. Table 2 presents a comparison of the models estimation of free surface shape close to the solid motion for an initial depth of 0.210 m.This table has of course less importance than the preceding one in terms of applications. Here again the results obtained by the models are satisfactory with almost all the predictions within the experimental resolution. We conclude that the Navier-Stokes models considered in this project are all able to model with a good accuracy the waves generated by a rigid body vertically falling into water taking into account the fluid/solid interaction. It is likely that the conclusion would be the same or not too different for a body entering water with an angle.

Figure 5. Solitary wave reflecting on a 2D vertical reef: case geometry (the tank is split in two parts for clarity: offshore on top, dyke side on bottom).

3 SOLITARY WAVE REFLECTING ON A 2D VERTICAL REEF

3.1 Case description

This benchmark aims at reproducing a set of experiments carried out at the O.H. Hinsdale Wave Research Laboratory, Oregon State University (OSU, see Roeber, 2010 and Roeber and Chung, 2012). These experiments involve the propagation, run-up, overtopping and reflection of high amplitude solitary waves on two-dimensional reefs. Their purpose is on one hand to investigate processes related to breaking, bore formation, dispersion, and passage from subto super critical flows, while providing, on the other hand, data for the validation of near-shore wave models in fringing reef.

The geometry of the test considered here is shown on Fig. 5. The length of the basin is of 104 m, however the computational domain is delimited by a reflecting wall at $x = 83.7$ m. The reef starts at $x = 25.9$ m with a nominal slope of 1/12. The actual slope is such that the height of 2.36 m is reached after 28.25 m. At this station a 0.2 m height crest is mounted. The offshore slope of the crest is the same as that of the reef, and the length of its plateau is of 1.25 m. The on-shore side has a slope of 1/15 giving a nominal length for the crest basis of 6.65 m (using the actual offshore slope, a crest basis of 6.64407 m is obtained). For the computations, the use of the nominal slope values is prescribed. This gives an offshore length of the crest slope (starting at 28.25 m) of 2.4 m.

The initial depth at still water is taken to be $h_0 = 2.5$ m, giving a partially submerged crest, and a depth behind it (on-shore side) of 0.14 m. The initial solution consists of a solitary wave of amplitude $A = 0.75$ m, giving a nonlinearity ratio of $A/h_0 = 0.3$.

To simplify the boundary conditions, the solitary wave is placed initially at $x = 17.6$ m which is in reality where the experimental data places the peak at dimensionless time $t\sqrt{(g/h_0)} = 47.11$. This shift will have to be applied consistently when comparing with experiments. Reflective walls are specified at both ends of the computational domain: $x = 0$ m and $x = 83.7$ m. The prescribed value of the mesh size is 0.05m. A Courant number of 0.4 is imposed. For the friction model, a Manning coefficient of 0.014 m $-1/3$ s is chosen. 3.2 Benchmarking results For this test numerical results are available from the following models: – THETIS (see Section 2.2), – EOLE (see Section 2.2), – TUCWave (Kazolea et al., 2012), developed by Inria and Technical University of Crete, solving hybrid Boussinesq/ Non-Linear Shallow Water Equations (NLSWE) with FV, – SLOWS (Ricchiuto & Filippini, 2014), developed by Inria, solving hybrid fully nonlinear GreenNaghdi/ NLSWE with a stabilized Finite Elements (FE) approach, – FUNWAVE-TVD (Shi et al., 2012), developed by Delaware University and Rhode Island University (US), solving hybrid fully nonlinear Boussinesq/NLSWE with structured meshes. Figs. 6a to 6d compare the measured and computed wave profiles for all numerical models as the numerical solitary wave propagates. Until the dimensional time $t* = 54.91$, the initially symmetric solitary wave propagates onshore, shoals across the toe of the slope at $x = 25.9$ m, due to the inclined bathymetry, and begins to skew to the front. As a result of shoaling the wave breaks around $t* = 59.3$ forming a plunging breaker. We can see that the numerical results of all models are different during the breaking process. TUCWave, SLOWS, and FUNWAVE mimic the breaker as a collapsing bore that slightly underestimates the wave height but conserves the total mass. EOLE andTHETIS overestimate the wave height, with a delay in the overall process observed with THETIS (though we do not know the behavior of the free surface between two measurement data points). The broken wave propagates on the back slope of the reef generating a super-critical flow moving into the stagnant region on the flat reef. While the bore propagates downstream a hydraulic jump develops at the back of the reef.The Boussinesq codes seem to capture this process in a stable manner. After the overtopping of the reef by the reflected bore, we can see the formation of an undular structure, whose inceptions are visible. As the water rushes down the fore reef, the flow transitions from flux to dispersion-dominated. These results show that both depth-averaged and fully three-dimensional models can predict these flows. In particular, the agreement between EOLE's, SLOWS'andTUCWave's results for most of the flow is quite surprising. Besides the resolution requirements

Figure 6a. Solitary wave reflecting on a 2D vertical reef:

free surface distribution at non-dimensional time = 66.53.

Top: Boussinesq-type models; bottom: Navier-Stokes codes.

Figure 6b. Continuation of Fig. 6a, non-dimensional time = 70.68.

Figure 6c. Continuation of Fig. 6b, non-dimensional

time = 76.33.

Figure 6d. Continuation of Fig. 6c, non-dimensional

time = 109.53. Figure 7a. Solitary wave reflecting on a 2D vertical reef: time series of the normalized free surface at wave gauge (WG) 2. Top: Boussinesq-type models; bottom: Navier-Stokes codes. Figure 7b. Continuation of Fig. 7a, WG 8. (definition of the interface), the precise definition of the quantities to be plotted may be in this case quite crucial for this type of flow, and should be related to the experimental uncertainty (unfortunately not available here). Figs. 7a to 7d compare the computed and recorded surface elevation time series at four specific wave gauges. The recorded data from the wave gauges at $x \leq 50.4$ m (i.e. WGs 2–8) show the effect of the dispersive waves on the free surface. The hydraulic jump developed at the fore reef produces a train of waves over the increasing water depth and the resulting undulations are intensified as higher harmonics

Figure 7c. Continuation of Fig. 7b, WG 10.

Figure 7d. Continuation of Fig. 7c, WG 13.

are released. All the Boussinesq-type models pro

vide a quite faithful description of the whole process,

including the dispersion dominated process observed

before the reef at later times. EOLE provides a good

description of the early phase of the process and a

somewhat satisfying prediction of the dispersion dom

inated undular bore, with higher frequency oscillations

missing in the results, perhaps due to a lack of reso

lution. THETIS provides a correct description of the

wave propagation, but it gives a late breaking incep tion with a phase advance of the bore on the reef

flat, and a phase lag of the reflected bore. In strongly

breaking regions THETIS data is also dominated by

very large oscillations which may be related to the

post-processing if strong air entrainment is predicted

by the VOF code. Figure 8. Impact on a urban area: plan view of the experiment in the basin and position of the offshore wave-gauges (up) and cross-section of the theoretical topography (down). Taken from Park et al. (2013). 4 SEASIDE EXPERIMENT: IMPACT ON A URBAN AREA 4.1 Case description This experimental case has been realized in the Oregon State University basin. A complex topography was built including a seawall and several buildings, inspired of the real city of Seaside (Oregon) at a 1/50 scale. The experiment consisted in generating a solitary wave with a piston-type wavemaker. This wave (height = 0.2 m, period = 10 s) was designed to correspond to the estimated tsunami wave height for the “500 years” CSZ (Cascadia Subduction Zone) tsunami. The experiment was reproduced 136 times, while displacing the sensors in the urban area (31 locations), and 99 of these trials have been judged as acceptable. As data about flows in an urban area remain very rare, this test-case is particularly interesting to compare the ability of the different models and of the different approaches to simulate the flows in a complex topography. An overview of the experiment is presented in Fig. 8, issued from Park et al. (2013). A LiDAR survey has been realized on the main part of the experimental basin (dimensions are 48.8 m long, 26.5 m wide and 2.1 m deep). BRGM has reconstituted an idealized topo-bathymetry based on this LiDAR survey. This idealized topo-bathymetry was used to fill the gaps in the original LiDAR data. Moreover, the topobathymetry was vertically moved for an initial water level at the 0-altitude, and horizontally limited to the area of interest. The proposed topo-bathymetry is represented in Fig. 9. Up, down and right boundaries are supposed to be walls, insofar as this case corresponds to a basin experiment (upand down-boundaries do not correspond to the basin limits, but available videos show that the urban area was closed by walls).The wave is generated

Figure 9. Impact on a urban area: view of the proposed

topo-bathymetry and positions of the sensors.

Figure 10. Impact on a urban area: time series of free sur

face elevation at four gauges. From top to bottom: gauges

WG3, B1, B6 and B9 (see Fig. 9). Black dotted lines: exper

iments by Park et al. (2013); light blue lines: EOLE; purple

lines: FUNWAVE-TVD; green lines: SURF-WB; orange

lines: Telemac-2D; brown lines: Calypso.

from the left boundary, using the water height measure

ments on WG1 of Fig. 9 (this implies that the reflected

waves are included in the signal). In the experiment,

the structure was constructed of smooth concrete with
Figure 11. Impact on a urban area: time series of velocity at three gauges. From top to bottom: gauges B1, B6 and B9 (see Fig. 9). Same color code as Fig. 10. The black circle indicates the estimated initial velocity. a flat finish, leading to an estimated roughness height of 0.1–0.3 mm. Consequently, a Manning coefficient of 0.010 m −1/3 s is suggested. 4.2 Benchmarking results Five codes were tested in this case: – EOLE (see Section 2.2), – FUNWAVE-TVD (see Section 3.2), – Telemac-2D, developed by EDF, solving NLSWE with FE or FV, – Calypso, developed by CEA, solving NLSWE with Finite Differences, – SURF-WB, developed by the universities of Bordeaux and Montpellier, solving NLSWE with FV. Fig. 10 shows time series of the free surface elevation for the five codes and measurements, on four of the gauges shown on Fig. 9. On gauge WG3, located offshore (middle of Fig. 9), all codes are late with respect to the data; this could be explained by a time error on this gauge in the available experimental data (for example, in Rueben et al., 2010, the peak on WG3 appears at about 20s, which is more consistent with the simulations). The delay seems to be larger with EOLE,

Figure 12. Impact on a urban area: 3D view with EOLE

(top) and first simulation with Sphynx (bottom).

the only code solving the Navier-Stokes equations in

this case. After the wave reflection on the coast, the experimental data show free-surface oscillations that are only predicted by FUNWAVE-TVD.

For gauges B1, B6 and B9 (located in the city on Fig. 9), all codes perform satisfactorily, reason ably well although differences with experimental data are significant, in particular at gauge B1. On the four gauges, EOLE (Navier-Stokes equations) and Telemac-2D (NLSWE) globally show the lowest dif ferences with data. On B1, the initial free-surface elevation peak is underestimated by all codes, but the subsequent behavior is correctly predicted except by FUNWAVE-TVD (ovestimation) and Calypso (under estimation). Similar conclusions can be drawn for B6. As for B9, the best models are EOLE and FUNWAVE TVD.

In estimating the consequences of tsunami impact, water levels are not sufficient. Fig. 11 shows velocity time series on the same gauges exceptWG3.The veloc ities are globally well predicted, even if in long-term Calypso tends to overestimate them and FUNWAVE TVD simulates a too important reflux near the coast line (in agreement with the abovementioned behavior regarding water elevation). New features under devel opment should improve the Calypso results especially on energy conservation. All codes have difficulties to predict the velocity on gauge B9, the most remote to

the initial coastline.

As a conclusion, the use of depth-averaged mod

els (such as NLSWE or Boussinesq-type models)

seems to be enough for such cases. Nevertheless, 3D

approaches may be interesting in calculating the result

ing force on buildings. Fig. 12 shows an example of

3D rendering with EOLE, along with a first simulation

Wave force calculations due to wave run-up on buildings: A comparison of formulas applied in a real case

F.G. Brehin, N. Zimmermann, V. Gruwez & A. Bolle

IMDC, Antwerp, Belgium

ABSTRACT: This paper presents the application of several empirical formulas for horizontal wave force calculations due to broken and/or overtopping waves for a series of beach profiles in North West France and illustrates the results of wave run-up, horizontal forces (F_h) and the limitations of the technique. The choice of the formulas from the literature review included the following: Goda-Takahashi, Camfield, Pedersen (USACE, 2006), Chen (Chen, 2012 and 2015), and a theoretical formula based on EurOtop (Pullen et al., 2007). Each formula is characterised by a particular application range, asymptotic behavior and specific input parameters which are discussed in detail in the paper.

Overall, results were characterized by large spatial variability with horizontal wave forces ranging from 0 to 100 kN/m, and the decrease of the force with distance from the crest. The greatest forces matched the areas

characterized by the largest wave overtopping volumes. Chen (2015) and EurOtop were applicable for most

of the profiles, and provided results within the range of average forces and intersected each other. Wave force

calculations based on the EurOtop formula gave reasonable force estimates, in the sense that despite their

theoretical character they matched with the empirical formula results of Chen (2012 and 2015) and were situated

in the average of all formulas.

1 INTRODUCTION

Existing formulas for pressure and force calculations due to waves were historically developed for the design of caisson breakwaters. Such formulas enable calcu lations of pressure on a submerged structure by a reflected or a breaking wave at the toe of the struc ture, in relatively large water depths. It is clear that these specific configurations cannot be representative of most waterfront configurations, where waves have already broken on a gently sloping dissipative beach where possible overflow concerns a shallow layer of water.

Similarly, several formulas were also developed for the stability of crown wall breakwaters due to wave run-up, which are typically located above the water line. This situation resembles more to the configura tion of the present study except for the major difference that a crown wall does not absorb all of the shock wave as it is low-rise and residual crossing flows are

present. Therefore this type of formulas underestimate

the pressure against an infinitely high wall in the same

configuration. On another hand, buildings are gener

ally located much more inland than crown walls and

breakwater configuration is not really representative

of typical beach profiles usually found in front of a

building in the coastal areas of France.

The formulas of greatest interest in this study are

mainly originating from the latest developments in this field, including the work of Ghent University and Flanders Hydraulics Institute, especially the thin bores impact breaking on the buildings. A theoretical inhouse formula was also proposed to transform run-up velocities and flow depths from the Eurotop guide (2007) into a force estimate. The translation of Eurotop’s run-up velocity and height into a force is purely theoretical, by analogy to a pressure force. Although this approach is sought for by the scientific community, reality is more complex. Since there is no solid literature on the topic, care has to be taken in interpreting the results of this formula, as they represent an order of magnitude with large uncertainties. First, it is crucial to recall the four types of wave impact: 1. Non breaking; 2. breaking (plunging) wave, almost vertical front; 3. Breaking (plunging) wave entrapping a large air pocket and 4. Breaking (runup) above sea level. Kortenhaus et al. (1998) presented an overview of the available formulas for wave force calculations on structures as part of the PROVERBS research project.This work contains the main formulas reported in the Coastal Engineering Manual (USACE, 2006; Chapter 5) and in the Rock Manual (CIRIA, 2009) and remains up to date. As part of a flood risk study in Northwest France, wave force calculations on schematized buildings located at a varying distance in the wave run-up zone were performed for a total of 33 measured profiles. The choice of the formulas from the literature review

included the following: Goda-Takahashi for caisson breakwaters, Camfield for beach run-up, Pedersen for crown walls (USACE, 2006), Chen for most recent wave run-up on emerged buildings (Chen, 2012 and 2015), and a theoretical formula based on EurOtop (Pullen et al., 2007). Each formula is characterised by a particular application range, asymptotic behavior and specific input parameters which are discussed in detail in the paper. All the above formulas enable the calcu lation of a horizontal force (F_h) except for Eurotop, for which the force was derived from the overtopping flow depths and velocities. Input parameters for the wave force calculations included both hydrodynamic and geometric data.

Care had to be taken in the interpretation of the wave force calculations for this project, as compared to classical applications of those formulas. In the present work, the goal was to determine the force acting on an infinitely high wall, representing a building close to the coastline which would be undergoing run-up flow and potential overtopping volumes. Profile geometry in front of this wall varied spatially but corresponded in most cases to a beach or gently sloping reef outcrop dissipating most of the wave energy before it reached the densely populated/developed areas.

An important step toward the regulatory translation of these results is the definition of adequate limits of resistance, depending on various parameters defining the works (exposure, material, structure, failure modes …). This translation, which was not the subject of this study, is far from obvious. Indeed normally a building is designed for a determined force, while here it is the opposite: given an existing structure, determine the upper bound force for which it was designed in order to identify where measures are needed. For these reasons it is suggested to define at national level the structural safety requirements based on a combination of science and common sense.

2 METHODOLOGY

2.1 Study area

The study area is located at the southern limit of the Armorica mountain range and is bordered by several dried marshes along the coastline which are flooded low areas characterized by more or less recent allu vial deposits. The geology and geomorphology is very variable/heterogeneous. There are two types of coastal sediments: muddy and sandy sediments from fluvial marine transport occur in the marshes and river mouths while aeolian sediment (coarser) dominate the beaches and dunes systems. The orientation of the shoreline is of importance for the area since it influences the hydrodynamic conditions (and especially the angle of

incidence of the approaching waves).

The area is characterized by large storm waves

and surge from the Atlantic Ocean. Previous stud

ies have shown extreme water levels in the range

of +4 m IGN69 for 100 yr. return period conditions (Alp'Géorisques and IMDC, 2014.The Sea Level Rise in the area is expected to vary between +40 cm and +60 cm over the next 100 years. (Alp'Géorisques and IMDC, 2014). The area is also characterized by numerous coastal structures (rubble mound at toe of dune, seawalls, etc.) which are important to determine for the force calculations. 2.2 Selection of formulas The selected formulas are as follow: – Goda (1985), amended by Takahashi (1994) for wave force calculations on a submerged caisson (quasi-static or impact wave), since this is the most popular for that type of use; – Camfield (1991) for wave forces due to run-up on a building located on a beach because this is the only historical formula incorporated in the USACE/Rock manual (excluding the recent work from Chen); – Pedersen (1997) for wave force calculations on crown walls (broken waves), because it is the most reliable formula for that particular type of application according to the Rock Manual (2009); – Chen (2012) and Chen (2015) for wave force calculations due to run-up on a building on a dike crest, because these are the only existing formulas for such applications; – Eurotop guide (2007), which does not provide a formula to estimate directly the horizontal force due to wave impact but enables the estimation of run-up velocity (v A) and height (h A) on both the seaward slope and dike crest. These parameters were translated into a theoretical pressure force F h per linear meter [N/m]: F h = v 2 A h A All the above formulas enable the determination of orders of magnitude only and it is important to recall that to quantify more accurately the forces, it is necessary to perform physical modelling or complex numerical modelling (CFD). Other formulas were not selected because of overlap with the selected formulas (one per application type was deemed sufficient), and some of them have also been considered too case specific. Additional works are in progress but remain unpublished, they have hence not been used for this study. There are many formulas because each force is dependent upon a particular geometry, type of wave impact and its impact time series. 2.3 Summary of conditions and range of applicability Each formula uses different hydrodynamic parameters and run-up formulas. Since these formulas were calibrated based on their specific test series, it is important not to harmonise (considering the same runup formula for example). The formulas of Pedersen, Table 1. Input parameters for the different formulas. Formula Basis Slope Run-up Goda Physical model Breakwater Max. height of impact Pedersen Physical model Breakwater 0.1% exceedance Camfield Theoretical Beach Max run-up Schüttrümpf 2.25*Hmax*tanh(0.5*ksi) Chen (2012) Physical model Dike Max run-up Schüttrümpf 2.25*Hmax*tanh(0.5*ksi) Chen (2015) Physical model Dike Max run-up Schüttrümpf 2.25*Hmax*tanh(0.5*ksi) Eurotop Theoretical Dike 2 % exceedance 1.65*Hs*ksi Table 2. Input parameters for the different formulas. Formula Free-board Distance Goda Water depth at toe of caisson set to 0 m From the breakwater crest (submerged berm) (usually negative, submerged structure) Pedersen Crest of breakwater From the breakwater crest Camfield Usually no freeboard, continuous slope From the shoreline Chen (2012) Dike crest From the dike crest Chen (2015) Dike crest From the dike crest Eurotop Dike crest From the shoreline

Table 3. Conditions used for generating the force envelope.

Formula Slope (tan α) Freeboard (m) Crest width (m)

Goda 1:0.5 to 1:4 (tested unknown) -5 m to -0.5 m 0 m to 25 m (tested 0.25 m to 20 m) (tested -3.9 m to -0.7 m)

Pedersen 1:1 to 1:3.7 (tested 1:1.5 to 1:3.5) 1.5 m to 5.0 m (tested 2.2 m to 3.8 m) 2.5 m to 7.5 m (tested 3.6 m to 7.2 m)

Camfield 1:10 to 1:100 NA NA (theoretical 1:10 to 1:100)

Chen (2012) 1:2 to 1:3 (tested 1:2.35) 1.5 m to 3.0 m (tested unknown) 0 m to 15 m (tested 0 m and 15 m)

Chen (2015) 1:1.5 to 1:8 (tested 1:3 to 1:6), 0 m to 2.5 m (tested 0.4 m to 1.5 m) 5 m to 25 m (tested 7.5 m to 22.5 m) foreshore slope: 1:35

Eurotop 1:1 to 1:10 (tested 1:1 to 1:8) 0 m to 5.0 m (tested unknown) NA

Camfield, Chen and Eurotop use different formulas

for run-up calculation (maximal run-up, 0.1% or 2%

exceedance run-up). The Goda formula uses the max imum impact level (η*) to replace the run-up that does not occur in that case. Particular care was taken in selecting the right type of wave input as well (Hs offshore, Hs toe of dike, Hmax).

The asymptotic behaviour was also determined to anticipate/provide results outside their application domain. Results obviously become informative only, for comparison purposes.

Goda : The formula is not slope dependent. It pre dicts a constant non-zero force to an infinite distance and a non-realistic behaviour for a positive freeboard. A condition/limiter was added to guarantee a positive force. In the present configurations, the formula is never applicable.

Pedersen :The formula is not defined in the case of a slope that is less than 1:3.7. For gentler slopes, the force was hence calculated with a slope of 1:3.7 (formula non-applicable), and the force tends to infinity in the case of a distance (from the crest) equal to 0. Camfield : The formula considers an infinite slope. This assumption was replaced by a constant force assumption after the crest to avoid underestimating the wave force further inland. Eurotop: The theoretical formula based on run-up velocity and height is the only one to experience a reasonable asymptotic behaviour for each of the following parameters: slope, distance and freeboard. Chen (2015): The force tends toward minus infinity if the distance (from the crest) is 0 (bad asymptotic behaviour of the conversion parameter C tr used in the formula). Therefore a limiter was added to control C tr . The waves are regular and defined at the toe of the dike. This was replaced by maximum significant wave Hmax = 1.86*Hs at a distance of 5*Hs of dike (and hence partially broken) for irregular waves. The validity limits for the generation of the force envelopes are summarized in Table 3, including the tested configurations (slope, freeboard and crest width) and therefore the actual validity domains presented in parentheses.

All the formulas had specific conditions imple

mented to set the force to zero when the freeboard

is larger than the maximum run-up height (no over

topping). In addition, the water level did not take into

account the set up due to the wave, because this was

implicitly included in the experimental results (like

for the overtopping volumes). The significant wave

heights at the toe of the dike corresponded to wave

propagation model outputs and were extracted at a

distance of 5*Hs of the toe of the dike.

2.4 Geometry data input

The beach profile data (x,y,z) used for the present

calculations correspond to input data for overtopping

calculations (Alp'Géorisques and IMDC, 2014). The

freeboard was the same as the one used for the dike

test. The slope was manually determined for each pro

file. It was generally different below water than on

the beach and dike. Since the slope influences wave

breaking, the slope underwater was preferred in case

of uncertainties. The structure depth for the Goda for

mula was manually determined for each profile at a distance equal to 5*Hs from the toe of the dike, where the wave height is defined. A steeper slope (more run up) or a lower freeboard led as expected to a stronger force.

2.5 Hydrodynamic data input

The hydrodynamic data consisted of wave and water level outputs from previous numerical model studies extracted for each profile (Alp'Géorisques and IMDC, 2014). Model scenario consist of the extreme storm event Xynthia (2010).

The water levels were extracted from a large scale circulation model (Delft 3D) with the wave data based on SWAN model outputs. The complete methodology was further validated in previous studies. A higher water level (lower freeboard) and higher wave heights led as expected to stronger forces.

3 RESULTS

An example of calculation outputs (Figure 1) shows the beach profile (bottom) along with the corresponding wave force distribution (top), the applicability range (thick lines) for each individual formula and the “most probable” force derived as the average of the applicable formulas (upper boundary of the red shaded area). The individual forces are represented by a dash line when they are not applicable and thicker line when they are.

If no force is defined, there is no envelope. This was the case for a few profiles in the study area that experienced too large calculated run-up values because of steep slopes. The distance is defined starting at the tip of the dike crest. The legend presents the values of run-up and surf parameter (ksi) calculated for each of the formulas. For the Goda formula, the run-up value corresponds to the maximum height of impact. In the absence of calibration data for these measured profiles, an expert judgement was made on the general behaviour of each formula and on the results relative to each other. It is worth mentioning that results of the different formulas were, even outside their application range, often within a factor 2 of the average results. This is remarkable given that a difference of an order of magnitude is easily found if insufficient care is taken of the exact definition of the input parameters. Results of the calculations for hydrodynamic conditions corresponding to the Xynthia Storm are presented for four profiles (profiles z2p1, z5p1, z12p2, and z16p1). Profile z2p1 (Figure 1, top left) presents an example of the applicability of the Chen formula (2015) and Eurotop, with a relatively steep slope (1:3.9) and a low freeboard (1.3 m). The plot illustrates well the domains of applicability of each formula. Goda and Pedersen formulas return incoherent values. The Camfield formula cannot be used because the slope is too steep. Forces calculated by the Chen formula are in the order of 30 kN/m close to the crest and decrease to 10 kN/m at the landward applicability limit. These values look within an acceptable order of magnitude and are confirmed by the Chen (2012) formula which calculates values at two points at the beginning of the crest and at a distance of 15 m (26 kN/m and 10 kN/m, respectively).The Eurotop formula is valid over the entire profile, and the overlap of results confirm that the theoretical formula chosen made sense despite its simplicity. Profile z12p2 (Figure 1, top right) is characterized by a steep slope and a wide and linear crest. This lies in the applicability domain of the Pedersen formula (rather limited – distance +2.5 m to +7.5 m past the crest). The force calculated by Pedersen varies from 45 to 25kN/m over the domain described above. On the first couple meters (distance from 0 to 2.5 m), the forces are way too large (hundreds of kN/m) due to the bad asymptotic behaviour of the formula. For Goda, the formula returns fairly realistic values (approximately 50 kN/m) even though the formula is not applicable because of the positive freeboard. The formulas of Chen (2015) and Eurotop yield values in the same range as those calculated with the Pedersen formula.This underlines the flexibility of these formulas, and the possibility to apply them over a fairly wide range of configurations. It is important to note that the curve force of Eurotop is below the curve of Chen (2015). A lower run-up value for Eurotop (4.9 m vs. 5.4 m for Chen) and the simple theoretical background could explain this difference. Overall force values were low (approximately 5 kN/m) at a distance starting 25 m after the crest and within the range of other forces. Profile z16p1 (Figure 1, bottom right) also illustrates the applicability of Pedersen, and Chen (2012 and 2015). The profile is characterized by a steep slope (1:2.3) although its relatively large freeboard (2.49 m) prevents large overtopping. The force estimation using the theoretical formulation based on Eurotop remains below the results from Chen (2012 and 2015). Profile

Figure 1. Calculated horizontal (F h) force distribution along profile (top) and associated beach profile characteristics

(bottom).

z5p1 (Figure 1, bottom left) presents an example of

applicability of Camfield formula with a gentle slope

(1:34) and a low freeboard (0.65 m), while all other

formulas are not applicable.

4 DISCUSSION

4.1 General discussion

The previous chapters have demonstrated spatial vari

ations in the horizontal wave force calculations on an

infinitely high wall (Fh) that occurred both along the stretch of coast but also along each profile. While such variations are obviously expected, variations between formulas for given settings should be limited. Such differences may be related to the exact assumptions and limitations in the use of formulas. These points are discussed in the paragraphs below. The interpretation of the results showed that it is impossible to generalize and choose one particular formula that would be adapted to calculate forces on the overall stretch of coastline (for our case and any other case in the world). It is therefore necessary to first determine the geometric characteristics of each profile, and then draw an expert judgement to state which formula is applicable

or not and which outputs have to be discarded. For

more reliable results, physical/and or numerical mod

elling (CFD) is a must, as it usually is for most coastal

structure design.

4.2 Applicability range of the formulas

Each formula uses specific hydrodynamic parame

ters. For example, run-up calculations correspond to

maximal run-up for the formulas of Chen (2012) and

Chen (2015), or the 0.1% and 2% exceedance run-up

for Pedersen and Eurotop, respectively. Run-up results

for Chen (2015) and Eurotop were fairly comparable

despite using the maximal run-up for Chen and the

2% run-up for EurOtop: This was compensated by the

different wave heights used in the formulas.

More variability was observed in the run-up cal

culated by Pedersen, which makes sense because this

formula was originally designed for a crown wall case

with a special configuration. Run-up calculations for

Camfield were overestimated for most of the profiles

mainly because of the lack of applicability because

of a too steep slope (two to three fold compared to other run-up formulas). The Goda formula, although it was never really applicable, matched surprisingly with the formulas applicable in the case characterised by a steep slope and a low freeboard, at least over the first twenty meters starting at the crest. The Chen (2015) and Eurotop formulas, although applicable only on a limited part of the profile, seemed to provide the most reliable wave force estimates. There was a good agree ment in the force calculations for those two formulas, with a comparable order of magnitude for both run up and force calculations. Results looked particularly realistic when forces for each formula overlapped. For steeper profiles (but > 1:1), the Pedersen formula gave satisfactory results, because the force at the top of the crown wall overlapped values from Eurotop and Chen (2015). It must be noted that Chen (2015) was only valid over a distance of 20 m starting at the crest. On the contrary, Camfield was only applicable for a gentle slope (1:10 to 1:100). The uncertainty in this formula is large because it is theoretical.

4.3 Uncertainties

The difference in the outputs of the different formu las gave a good idea of the uncertainties within the results. Uncertainty was fairly large and ranged in the order of ±30% of the calculated envelope. For this reason, a detailed sensitivity analysis was not con ducted. Additional uncertainties could be due to the fact that calculations did not take into account the profile evolution during the storm event and the con tinuous erosion of the crest. In reality, some profiles corresponded to a dune system with rubble mound or rock layers at the toe for stabilization. In order to match real conditions it would have been ideal to apply the formulas to the post-storm profiles and not the exist ing profiles. Nevertheless this aspect is not expected to modify the present conclusions since each formula already implies a simple schematisation of a strongly variable profile.

4.4 Recommendations for horizontal wave force calculations on an infinitely high wall

Recommendations that were learned from this study include determining at first the limits of applicability of the formulas for each profile, and inputting the proper hydrodynamic and geometric conditions. Globally the formula of Chen (2015) and Eurotop provided reasonable estimates for profiles characterized by slopes comprised between 1:1 and 1:10 and freeboards between 1 m and 3 m. It was noted that the Eurotop formula was quite flexible and offered a validity domain that overlapped the major part of the profile (as compared to other formulas). However, care should be taken in the interpretation and use of the Eurotop force estimation because it is only related to the formulas for run-up velocity and run-up depth calculations. Even if those two values are correct, the theoretical basis of this conversion formula into a force is questionable, because it has not yet been validated by the scientific literature. For slopes milder than 1:10 the Camfield formula is recommended to estimate the force at the beginning of the crest. In general, the results were comparable to the strength values calculated for other projects that determined wave forces on walls (IMDC, 2012 & 2014).

5 CONCLUSIONS

This study enabled the calculation of horizontal wave forces on an infinitely high wall over a series of profiles on the coast of North West France. Maximum force values were in the order of a few hundreds of kN/m at the beginning of the crest, and corresponded to the profiles with low freeboards and large overtopping volumes. There are numerous uncertainties that can have a significant impact on the forces and must be taken into account. They originate from the formula itself (calculation or wrong range of applicability), but also from uncertainties due to expert judgement of the geometric parameters. Results showed that it was impossible to generalise and choose a particular formula to calculate forces over an entire coastline (with complex morphology) such as the study area because each formula was defined for a very specific case. It is necessary to first determine the geometric characteristics of the profiles and perform an expert judgement about the range of applicability of each formula and determine the outputs to discard. In order to compensate for that limitation, an envelope force was defined. However, there were still some profiles where only the theoretical formula based on the Eurotop guide was applicable. Therefore there are still many gaps in the force calculation knowledge. The force envelope can be used as a basis

for regulatory requirements, taking into account the

significant uncertainty associated.

More specifically, for the scientific aspect, the study

of the wave force calculation formulas highlighted the

following:

- The Goda formula was presented for information

only because in reality it is never applicable for any

profile because it considers a negative freeboard.

- The Pedersen formula is applicable for the case of

a steep slope (1:1 to 1:3.5) and has a very limited

applicability range (distance of 5 m). It does not

enable to calculate forces at distances past that dis

tance but can provide a check with other formulas

(Chen, Eurotop for example).

- The Camfield formula, based on theory, can give a first indication of forces for slopes below 1:10 (1:10 to 1:100).

- The Chen (2012) formula calculates the force at two distinct points (at a distance of 0 and 15 m from the crest) which does not enable the determination of the force evolution as a function of distance but enables a good comparison with Chen (2015) and Eurotop.
- The Chen (2015) formulas, although their applica bility range covers only a part of the profile, seem to provide the best force estimates for slopes between 1:1 and 1:10.
- The theoretical pressure force proposed based on the Eurotop guide is a good indicator of orders of magnitude to expect despite its simplicity. It is particularly appealing in settings with strong hydraulic and geometric variations, because of its good asymptotic behavior. This cannot be said of most formulas found so far in literature. It is therefore strongly suggested that future research pays attention to the asymptotic behavior of new formulas proposed.
- These values remain approximations. They cannot replace a thorough physical or numerical modelling study.

Alp'Géorisques and IMDC. 2014. Plan de Prévention des

Risques Littoraux en Vendée. DDTM 85. Rapport de

Phase 1.

Alp'Géorisques and IMDC. 2014. Plan de Prévention des

Risques Littoraux en Vendée. DDTM 85. Rapport de

Phase 2. Cavanagh, C, Webb W.F., and Hodgkinson, H.R. 1982. Insitu lateral load tests on an old brickwork building. British Ceramic ResearchAssociation Ltd.Technical Note No 345. Chen, X., Hofland, B., Altomare, C., Suzuki, T., and Uijttewaal, W. 2015. Forces on a vertical wall on a dike crest due to overtopping flow. Coastal Engineering 95, 94-104. Chen, X., 2011. Hydrodynamic loads on buildings caused by overtopping waves. Master Thesis for Delft University of Technology, Delft, Netherlands; Project number: 770_59 for Flanders Hydraulics Research, Antwerp, Belgium. CIRIA, CUR & CETMEF, 2007. The Rock Manual. The use of rock in hydraulic engineering (2nd edition). C683, CIRIA, London. FEMA, 2011. Coastal Construction Manual. Principles and Practices of Planning, Siting, Designing, Constructing, and Maintaining Residential Buildings in Coastal Areas, Fourth Edition. FEMA P-55/Volume I/August 2011. IMDC, 2012. Bepaling van de golfkrachten op de dijk van de tijdelijke stop van de aansluiting naar de westelijke havendam. I/NO/12213/12.162/VGR/. IMDC, 2014. Alpha Island BOG. Wave forces on a vertical wall due to wave overtopping. I/NO/11431/13.495/SUD/. Kortenhaus, A., Oumeraci, H., Allsop N. W. H., McConnell K.J., Van Gelder, P.H.A.J.M, Hewson P.J., Walkden M., Muller G., Calabrese M., and Vicinanza, D. 1998. MAST III/PROVERBS. Chapter 5.1. Wave impact loads - Pressures and Forces - PROVERBS (Probabilistic Design Tools forVertical Breakwaters), contract MAS3-CT-0041. Final Report Pullen, T., Allsop, N.W.H., Bruce, T., Kortenhaus, A., Schüttrumpf, H. & Van der Meer, J.W. 2007. EurOtop. Wave overtopping of sea defences and related structures: assessment manual. CIRIA, CUR & CETMEF (2007). The Rock Manual. The use of rock in hydraulic engineering (2nd edition). C683, CIRIA, London. Takahashi, S., Tanimoto, K., and Shimosako, K. 1994. A proposal of impulsive pressure coefficient for design of composite breakwaters. Proc. Of Int. Conf. on HydroTechnical Eng. For Port and Harbour Construction. Port and Harbour Research Institute, Yokosuka, Japan. U.S. Army Corps of Engineers. 2006. Coastal Engineering Manual. Manual 1110-2-1100, U.S.A.C.E, Washington, D.C. (in 6

volumes). Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Coastal tsunami-hazard mapping

Cuneyt Yavuz & Elcin Kentel

Department of Civil Engineering, METU, Ankara, Turkey,

ABSTRACT: Mediterranean experienced numerous earthquakes and some of them triggered tsunamis through

out the history. Historical records, locations of fault zones and volcanoes indicate that there are some tsunamigenic

sources in the Mediterranean Sea. Thus, tsunami hazard assessment of the region is important for designing early

warning systems, site selection of future critical infrastructures (CIs) and planning necessary mitigation measures

for existing CIs and settlement areas. In this study, a probabilistic analysis of tsunamis is conducted along the

Cyprus arc. Fitted probability density functions (PDFs) of the focal depth and the magnitude are used together

with source parameters of historical earthquakes to generate PDFs of inundation depth at a number of CIs along

the Gulf of Mersin. Locations of the simulated earthquakes are selected from the previously recorded tsunamis

originated in the Mediterranean Sea and NAMI-DANCE software is used to estimate inundation depths. Selected

CIs include ports, industrial zones, and waste water treatment plants. Simulated run-up results are assessed and

PDFs of inundation depths at the CI locations are analyzed. It is concluded that some of the existing CIs may be

vulnerable to probable future tsunamis.

Keywords: Tsunami, probabilistic tsunami analysis, critical infrastructures, NAMI-DANCE

1 INTRODUCTION

The term “Tsunami” which is composed of Japanese words “Tsu” and “nami” meaning harbor and wave, respectively is generated after The Great Meiji San riku Tsunami occurred on June 15, 1896 at the coast of the Tohoku region (Nakao, 2001). Tsunami is a series of seismic sea waves generated by different types of natural hazards. Landslides, volcanic eruptions, glacier calvings, meteorite impacts and mostly under sea earthquakes can be potential hazards to trigger a tsunami. If an earthquake occurs under the sea, it cre ates a displacement and from this rupture, huge amount of energy is transferred to the water column over the displaced area. This enormous amount of energy causes a sea level rise and generates a tsunami. Histor ical tsunami events show that densely populated areas along the coastline and coastal structures as harbors, marinas, aquaculture areas may be severely affected by tsunamis. In recent years, several destructive tsunamis were experienced in Indonesia (2004), Samoa (2009), Chile (2010) and Japan (2011). After the Boxing Day Tsunami inAceh province, Sumatra, Indonesia (2004), “Tsunami Risk Reduction” has become one of the most significant issues for governments throughout the world (Lovholt et al., 2012).

Although, tsunamis are generally observed in the

oceans and open seas, many tsunamis have been

recorded in the Mediterranean Sea due to the nat

ural hazards in this region. The Mediterranean Sea

is approximately 3900 km long, its maximum width is around 1600 km and greatest depth is 4400 m. It is one of the biggest marginal seas on the planet. During the last 36 centuries, 67 earthquakes with magnitudes greater than 7 ($Mw > 7$), 133 earthquakes with magnitudes between 6 and 7 ($7 > M w > 6$) and at least 96 tsunamis were documented in the Eastern Mediterranean (Altinok and Ersoy, 2000). Not all earthquakes generate tsunamis. The University of The West Indies Seismic Research Centre states that four conditions should be satisfied for tsunami generation (Web1): (1) The earthquake must occur beneath the ocean or cause material to slide in the ocean. (2) The earthquake must be strong, at least magnitude 6.5. (3) The earthquake must rupture the Earth's surface and it must occur at shallow depth – less than 70 km below the surface of the Earth. (4) The earthquake must cause vertical movement of the sea floor (up to several meters). Identification of tsunamigenic earthquakes through tsunami early warning systems is a challenging task. Effective early warning systems require knowledge and assessment of risk (Basher et al., 2006; UN/ISDR, 2006). The impact of a tsunami can be mitigated through a reliable early warning system. Timely and reliable warning is necessary so that those responsible for responding to the warning will feel confident taking necessary actions (Grasso and Singh, 2011). Otherwise, reaction of the community and

local governments gradually reduce after false alarms.

Additionally, money will be wasted to take unneces

sary precautions like evacuation of the population to

safe locations (Blackford and Kanamori, 1995).

Probabilistic tsunami assessments should be con

ducted to analyze likely adverse effects of tsunamis

along the specified region using numerical modelling

tools. Different numerical models have been proposed for tsunami simulations since 1960’s. Some of the most commonly used Tsunami modelling softwares are TUNAMI N1 (Imamura, 1995), MOST (Titov, 1999), NAMI DANCE (Zaytsev andYalçıner, 2006) and SIFT (NOAA, 2015). In this study, NAMI-DANCE is used to simulate tsunamis originating in the selected criti cal zones of the Mediterranean Sea. NAMI-DANCE uses finite difference method to solve non-linear shallow water wave equations to analyze and com pute generation, propagation and amplification of selected tsunamis depending on the source parame ters using seismic rupture characteristics. The model computes all components of tsunamis in shallow waters and the inundation zones along the specified coastline.

Geist & Parsons (2006) worked on the effective ness of probabilistic seismic hazard analysis (PSHA) and probabilistic tsunami hazard analysis (PTHA) on the evaluation of historical tsunami run-up data. They mentioned that tsunami run-up heights are affected from some of the earthquake source parameters and PTHA may become very complex and difficult to carry out. They proposed that empirical and computational methods can be integrated through Monte Carlo simu lations to analyze tsunami hazards. They selected two different regions and demonstrated their method in these regions. In a more recent study, Gonzalez et al. (2009) developed inundation maps using the correla tion between probabilistic tsunami hazard analysis and probabilistic seismic hazard analysis. In the current study, Monte Carlo analysis is used to propagate uncer tainties associated with variable source parameters to inundation levels at the coast.

The aim of this study is to conduct a series of tsunami simulations originating at a critical tsunami source zone located in the East Mediterranean Sea using fine bathymetry data. NAMI-DANCE Tsunami Simulation Software is used to determine probabilis tic wave amplitudes at a number of selected critical location at the Gulf of Mersin, (Turkey). Impacts of tsunamis on the current settlement areas and CIs along the cost are evaluated and useful information for site selection of future CIs is generated.

2 TSUNAMI MODELLING PROCEDURES

Accurate bathymetry data and reliable rupture param eters are needed to generate a coastal tsunami and conduct hazard mapping of the selected area. Also, reliability of the necessary data should be examined by modelling the historical tsunamis and comparing the results of the actual hazards and the numerical models.

NAMI-DANCE uses the following non-linear shallow water wave equations in tsunami modeling: where M is the discharge flux in x-direction, N is the discharge flux in y direction, D is the water depth, g is the gravitational acceleration, is the water surface elevation and k is the bottom friction coefficient. Geller (1976) worked on scaling rules for earthquake source parameters and derived some equations using 41 historical earthquakes. He used some observational and empirical data sets to define the length, L, and the width, W, of the rupture and found that L = 2W. Kanamori and Anderson (1975b) derived an equation to define the fault length L, depending on seismic moment, M_0, of an earthquake: M_0 can also be used to define the displacement amount due to an earthquake as follows: where μ is the shear modulus of the crust (in Pa) and D is the displacement (in m). Generally, Richter magnitude, M_w, of the earthquakes are provided in the literature.Therefore following equation should be used to define seismic moment from Richter magnitude of the earthquake provided by National Oceanic and Atmospheric Administration (Web2): Using these formulas, source parameters can be scaled depending on the magnitude of the earthquake. In this study, probabilistic analysis for a number of earthquake parameters are conducted using the historical earthquake data collected in the Mediterranean Sea. Historical tsunamigenic earthquakes and related parameters recorded in the Mediterranean Sea are collected and documented in EU funded TRANSFER project (TRANSFER, 2009) and online KOERI Earthquake Archive. In addition, Tsunami hazard database provided by Necmioglu (2014) lists more than 5000 tsunamigenic source parameters observed along the Turkish coastlines between 1911 and 2011. Within the selected region, a map of unit tsunami sources through the Hellenic and Cyprus Arcs are determined using ComMIT interface (Figure 1). Hellenic and Cyprus arcs are divided into 20 subzones to inspect tsunamigenic earthquakes along these

Figure 1. Unit tsunami sources through the Hellenic and

Cyprus Arcs using ComMIT interface (ASTARTE, 2015).

Figure 2. Frequency diagrams of focal depth, and magni

tude of the tsunamigenic earthquakes observed in the selected

part of the Cyprus Arc.

faults in the EU fundedASTARTE project (ASTARTE, 2015). A set of simulations is conducted to identify the critical source location for the CIs located at the Gulf of Mersin (marked with a black square in Figure 1) and sub-zone 18 is selected as a result of these analysis. Thus, historical earthquakes originated within sub zone 18 are considered as potential tsunami sources in this study.

2.1 Probabilistic Analysis of Historical Tsunamis in the Mediterranean Sea

Within the scope of this study, CIs and populated areas only along the Gulf of Mersin are examined.Therefore, 24 critical structures are identified along the Gulf of Mersin. Tsunamigenic earthquake catalog provided in the TRANSFER project (TRANSFER, 2009) together with KOERI earthquake archive (Web3) are used to identify historical earthquakes observed in sub-zone 18. TRANSFER project catalog and KOERI earth

quake archive include all the historical earthquakes Figure 3. Cumulative distribution functions (the bold lines) of Richter magnitude and focal depth. occurred between 1907 and 2016. In these two catalogs a total of 23 earthquakes are identified within sub-zone 18. NAMI-DANCE requires a set of earthquake source parameters including longitude and latitude of the earthquake, magnitude, focal depth, strike angle, dip angle, rake angle, fault length and fault width. Magnitude and focal depths of the selected 23 earthquakes taken from TRANSFER and KOERI catalogs are used in the probabilistic analysis. Probability density functions providing best fits for the magnitude and focal depths are found. Frequency diagrams of the focal depth and magnitude for the selected 23 earthquakes are provided in Figure 2. As can be seen from Figure 2, the Richter magnitudes of the selected earthquakes range between 3 and 6 while the focal depth ranges between 0 and 35 km. As the second step, best fitting distributions for the magnitude and focal depth are identified. Lognormal distribution and normal distribution are fitted for the focal depth and magnitude of the earthquakes, respectively. Cumulative distribution functions (CDFs) for these two variables are provided in Figure 3. Mean and standard deviations for the magnitude are 4.59 and 0.63 while they are 13.49 and 5.89 for the focal depth, respectively. Since the other source parameters as strike, dip and rake angles are only dependent on the structure

of the fault, probabilistic distributions are not fit

ted to these parameters. Instead, these parameters are

directly obtained from the source database prepared

by Necmioglu (2014).

2.2 The Study Area

Mersin is located in the Southern part of Turkey. It is

one of the most prominent touristic destinations in the

country and has many attractive natural and historical

sites. There are lots of summer houses and CIs such as

a water treatment plant, marinas, an international port

etc. along the Gulf of Mersin. Study area marked on

the bathymetry map and detailed Google Earth satel

lite image can be seen in Figure 4(a) and Figure 4(b),

respectively.

A detailed search is conducted using the litera

ture and Google Earth satellite images to identify CIs

located along the coastline between Mersin Province

and Erdemli (red rectangle in Figure 4). A total of

24 CIs are identified and listed in Table 1. In order

Figure 4. (a) Bathymetry map. (b) Google Earth satellite

image of the study area.

Table 1. Critical Infrastructures located along Mersin-Erdemli coastline.

Gauge No	Name	City	Altitude (m)	Latitude	Longitude	Max. (+)ve amp.(m)
1	Soda Chemical Industry	Mersin	0	34.73	36.81	0.75
2	Karaduvar W.T.P.	Mersin	0.5	34.71	36.81	0.99
3	Karaduvar City Centre	Mersin	0	34.69	36.81	0.97
4	Mersin Free Zone 1	Mersin	0.7	34.67	36.81	0.95
5	Mersin Free Zone 2	Mersin	-0.3	34.66	36.80	0.83
6	Mersin Free Zone 3	Mersin	-0.3	34.65	36.80	0.83
7	Mersin International Port	Mersin	-0.3	34.64	36.80	0.83
8	Mersin Congress Centre	Mersin	13.2	34.63	36.80	0.00
9	Mersin Marina	Mersin	1.8	34.63	36.79	0.89
10	Mersin_Funfair	Mersin	0.8	34.63	36.79	0.89
11	Tevfik Sırrı Gur Stadium	Mersin	1.1	34.62	36.79	1.35
12	Mersin Hilton	Mersin	1.1	34.61	36.78	1.35
13	Waterside Park	Mersin	14.5	34.59	36.78	0.00
14	Mersin Yacht Marina	Mersin	2.5	34.57	36.77	0.00
15	Mersin Mezitli 1	Mersin	0.7	34.54	36.75	1.12
16	Mersin Mezitli 2	Mersin	8.7	34.53	36.74	0.00
17	Mersin Mezitli 3	Mersin	36.9	34.48	36.71	0.00
18	Mersin Summer Houses1	Mersin	-0.4	34.46	36.70	0.77
19	Mersin Summer Houses2	Mersin	25.4	34.46	36.70	0.00
20	Mersin Summer Houses3	Mersin	5	34.38	36.65	0.00
21	Mersin Summer Houses4	Mersin	5	34.37	36.64	0.00

22 Erdemli Yacht Marina Mersin 0.1 34.32 36.61 0.70

23 Erdemli Coastline Mersin 26.5 34.30 36.60 0.00

24 METU-IMS Mersin 7.8 34.26 36.56 0.00 to determine run-up levels at these CIs, a gauge is assigned in the NAMI-DANCE software to each of these locations. 2.3 Data Processing for the Study Area In this study, 270 m grid size high resolution bathymetry is produced using GEBCO © (General Bathymetric Chart of the Oceans) software. Then the contour map is digitized using SURFER © version 12 and gauges are assigned in NAMI-DANCE at 24 CI locations (see Figure 5). 2.4 Simulation and Propagation of Selected Tsunamis The source parameters like strike, dip and rake angles used in this study are provided in Table 2. Location of the simulated earthquakes are randomly selected Figure 5. Gauge locations representing CIs along Mersin-Erdemli coastline.

from the listed historical earthquake locations given

in Table 2.

Simulations are conducted for 23 earthquakes and

corresponding inundation depths are defined for 24

gauges. NAMI-DANCE software takes source and

source parameters such as longitude and latitude of

epicenter, dip angle, strike angle, focal depth, mag

nitude and displacement as inputs and generate all

necessary tsunami propagation parameters such as

maximum and minimum sea level changes, run-up

levels etc. as outputs.

Magnitude and focal depth of the test cases are

sampled from the fitted probability density functions

of these parameters and Monte Carlo simulations are

conducted using data provided in Table 2. Maximum

and minimum wave propagations for an example sim ulation are given in Figures 7a and 7b, respectively. Source parameters of this example simulation are pro vided in Table 3 and the location of the source is given in Figure 6.

3 DISCUSSION and CONCLUSIONS

Probabilistic tsunami hazard analysis is performed for the Gulf of Mersin. A limited number of Monte Carlo simulations is realized. Gauges are assigned to the locations of the critical infrastructures along the Gulf of Mersin and positive wave amplitudes are computed at all the gauges. Tsunami simulation durations are selected as 6 hours for each run.Tsunami run-up levels are determined from 23 Monte Carlo simulations and maximum positive wave amplitudes are given in the last column of Table 1. For demonstration purposes, inundation depth distributions at two critical gauges,

Table 2. Tsunami Source Parameters for the selected 23 earthquakes.

Latitude	Longitude	Strike (0)	Dip (0)	Rake (0)	Richter Magnitude	Depth (km)
36.26	34.89	236	52	70	4.7	8.0
33.74	35.19	213	60	35	5.5	13.3
33.05	34.58	220	54	70	3.4	2.3
35.99	34.58	232	52	70	4.5	1.0
35.25	35.65	234	52	70	3.8	6.6
36.03	35.89	215	55	60	3.9	15.2

32.20 35.50 196 58 256 5.1 6.6

36.23 35.28 214 54 60 5.5 3.3

36.30 34.33 163 77 255 6.4 7.7

36.72 36.03 221 40 309.5 4.3 4.1

33.70 35.50 224 52 60 4.4 14.1

35.29 35.32 269 36 71 5.3 18.1

35.55 35.25 320 32 106 4.4 13.5

36.18 35.89 108 90 90 5.0 14.0

34.10 35.50 237 43 64 3.3 11.3

33.85 35.73 264 72 341 4.4 4.8

34.63 33.86 312 19 96 4.1 13.2

31.50 35.50 320 32 106 3.6 19.1

36.51 35.07 338 28 106 4.9 23.7

36.24 35.92 40 49 249 4.8 11.0

34.74 34.41 60 60 268 4.6 3.9

35.20 35.60 74 28 100 3.7 14.5

35.00 33.90 110 72 83 5.3 15.7 namely Mersin Stadium and Mersin Free Zone 1, are given in Figure 8 and Figure 9, respectively. As can be seen in Table 1, up to 1.35 m inundation depths are observed at the selected CI locations. As a result of only a limited number of Monte Carlo simulations greater than 1 m tsunami waves are observed along the coastline of the Gulf of Mersin. In order to generate more realistic results, number of simulations need to be increased. This will allow sampling from the whole ranges of magnitude and focal depth and generation of realistic inundation depth frequency distributions at critical locations. Table 3. Source parameters of the sample case (M w = 6.4). Epicenter 34.33 0 E Dip Angle (0) 77 36.3 0 N Strike Angle (0) 163 Rake Angle (0) 255 Focal Depth (km) 7.7 Displacement (m) 6 Length of fault (km) 46 Width of fault (km) 23 Figure 6. Source of the sample tsunami.

Figure 7. (a) Maximum wave propagation in the study area for the sample tsunami (b) Minimum wave propagation in the study area for the sample tsunami.

Figure 8. Positive wave amplitude on Mersin Stadium.

Figure 9. Positive wave amplitude on Mersin Free Zone1.

As future research it is planned to obtain depth damage curves of different CIs and to estimate overall risks using probabilistic inundation depths generated as a result of this study together with depth-damage curves.

Altınok,Y. and Ersoy, S¸. 2000.Tsunamis observed on and near Turkish coast, Natural Hazards,Vo1.21, No 2–3, 185–205. Doi:10.1023/A:1008155117243

ASTARTE. 2015. Deliverable D3.12,ASTARTE-Assessment, STrategy And Risk Reduction for Tsunamis in Europe. Grant 603839, 7th FP (ENV.2013.6.4-3 ENV.2013.6.4-3).

Audet, C., Couture, G., Dennis Jr, J. E., Abramson, M. A.,

Gonzalez, F., Titov, V., & Spillane, M. 2005. Optimal placement of tsunami warning buoys using mesh adaptive direct searches. Optimization days, Montreal, May. Available at: https://www.gerad.ca/Charles.Audet/Slides/ Tsunami_JOPT Basher, R. 2006. Global early warning systems for natural hazards: systematic and people-centered. Phil.T. Roy. Soc. A., 364, 2167–2182. Doi: 10.1098/rsta.2006.1819. Blackford, M., Kanamori, H., Tsunami Warning System Workshop Report, NOAA Tech. Memo. ERL PMEL-105 (PB95-187175), 95 pp, (September 14–15, 1995). Community Model Interface for Tsunami (ComMIT), NOAA Center for Tsunami Research, PMEL. Available at: http://nctr.pmel.noaa.gov/ComMIT/ Geller R. J. 1976. Scaling Relations for Earthquake Source Parameters and Magnitudes, Bulletin of the Seismological Society of

America. Vol. 66, No. 5, pp. 1501–1523. Geist E., L. and Parsons T. 2006. Probabilistic Analysis of Tsunami Hazards, Natural Hazards vol.37 277–314 Doi: 10.1007/s11069-005-4646-z Gonzalez F.I., Geist E.L., Jaffe B., Kanog˘lu U., Mofjeld H., Synolakis C.E., Titov V.V., Arcas D., Bellomo D., Carlton D., Horning T., Johnson J., Newman J., Parsons T., Peters R., Peterson C., Priest. G.,VenturatoA., Weber J., Wong F., Yalciner A.C. 2009. Probabilistic tsunami hazard assessment at Seaside, Oregon, for near and far-field seismic sources, Journal of Geophysical Research, vol.114, Doi: 10.1029/2008JC005132. Løvholt, F., Glimsdal, S., Harbitz, C.B., Zamora, N., Nadim, F., Peduzzi, P., Dao, H. and Smebye H. 2012. Tsunami hazard and exposure on the global scale, Earth-Science Reviews, Vol 110, Issues 1–4, pp.58–73, January 2012. Doi: 10.1016/j.earscirev.2011.10.002. Kanamori, H. and D. L. Anderson (1975b). Theoretical basis of some empirical relations in seismology, Bull. Seism. Soc. Am. 65, 1073–1095. Necmioglu O. 2014. Tsunami Hazard In Turkey And Surroundings, Ph.D. Thesis, Istanbul Technical University,Istanbul,Turkey Nakao, M. 2001. The Great Meiji Sanriku Tsunami, Failure Knowledge Database/100 Selected Cases, Available at: http://www.sozogaku.com/fkd/en/hfen/HA1000616.pdf NAMI DANCE, 2011, Manual of Numerical Code NAMI DANCE, published in http://namidance.ce.metu.edu.tr TRANSFER. 2009. Deliverables of EU Funded TRANSFER (Tsunami Risk and Strategies for European Region), Available at: http://www.transferproject.eu, last accessed on December 2014. UN/ISDR. 2006. Global survey of early warning systems. United Nations International Strategy for Disaster Reduction, Geneva, 56 pp. Web1: Seismic Research Centre. 2016. “Do all earthquakes cause tsunamis?” The University of The West Indies. Available at: http://www.uwiseismic.com/General.aspx?id= 21#allearthquakes Web2: National Oceanic and Atmospheric Administration (NOAA). Measuring Earthquake Size, Available at: http:// nctr. pmel .noaa.gov/education/ITTI/usgs/GPSthailand/lecture4. pdf Web3: Boun KOERI Regional Earthquake-Tsunami Monitoring Center, Available at:http://www.koeri.boun.edu.tr/ sismo/2/en/ Yalciner A. C., Dilmen I. D., Insel I. 2008. An Investigation on the Tsunami Risk Map of Turkey, Report of the Project ODTÜ, 06-03-03-2-02-05, for Ministry of Transport, Turkey. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Waves generated by ship convoy: Comparison of physical and numerical modeling with in-situ measurements

T. Cohen Liechti, G. De Cesare & A.J. Schleiss

Laboratory of Hydraulic Constructions (LCH), Ecole Polytechnique Fédérale de Lausanne (EPFL), Lausanne,

Switzerland

R. Amacher

Laboratory of Applied Mechanics and Reliability Analysis (LMAF), Ecole Polytechnique Fédérale de Lausanne

(EPFL), Lausanne, Switzerland

M. Pfister

Filière de génie civil, Haute école d'ingénierie et d'architecture (HEIA-FR, HES-SO), Fribourg, Switzerland

ABSTRACT: A part of the domestic waste of the city of Geneva (Switzerland) is transported with ship convoys

on the Rhone River to the waste incineration station. These convoys generate waves, which partially endangers

the stability of the river banks and the riparian fauna. To reduce the dominant wave peaks, a flap was added at

the stern of the barge. The efficiency of that flap was tested in physical and numerical model tests, and then

compared to in-situ measurements. This case study focuses on a discussion of the appropriateness of the two

models, by describing their accuracy for the present case. It indicates that the physical model reproduces the

wave heights almost correctly, but does not re-produce adequately the dominant frequencies. In contrast, the

numerical model damps the wave heights significantly, but gives correct dominant frequencies.

1 INTRODUCTION

Since several decades, a part of the domestic waste of

the City of Geneva (Switzerland) is transported on the Rhone River from the city center to the waste inciner ation station outside of the city with ship convoys con sisting of a pusher tug and a barge. The pusher tug has a length of 12.1 m, a width of 5.5 m, and a weight of 52 tons; The barge is 43.0 m long, 8.6 m wide, and weighs 120 tons. The transport capacity of a barge is 170 t. Waves generated by these convoys may damage the riverbanks and affect the riparian fauna (Nanson et al. 1994, Coops et al. 1996, Bishop 2003, De Roo et al. 2012), requiring consequently protection and mainte nance measures. As an efficient approach, a reduction of the convoy velocity is frequently discussed, partic ularly as the convoy passes a nature reserve. The latter is, however, not appropriate for logistic reasons, so that adaptions on the hull are considered by the Industrial Services of Geneva (SIG) as operator. To specify such adaptions, and to quantify their effect, SIG assigned the Laboratory of Hydraulic Constructions (LCH) of Ecole Polytechnique Fédéral de Lausanne (EPFL) to propose related options and to validate them with phys ical and numerical model tests.The latter indicated that a flap mounted at the barge stern is effective (Fig. 1), as the waves generated between the pusher tug and the barge are critical. Those are significantly reduced by
Figure 1. Sketch of pusher tug (left) and barge (right),

with wave-reducing flap at barge stern. the flap, as the pusher tug and the barge are hydrodynamically linked. The flap is operated by hydraulic cylinders and lowered during journey, but lifted up in the port to facilitate manoeuvers. SIG owns one pusher tug and four barges, of which one was equipped with the recommended flap. In order to verify its efficiency, SIG appointed LCH with in-situ measurements of the waves generated by two types of barges: (1) modified including the flap, and (2) original, not modified barge without flap (Amacher et al. 2015). To complete the investigation, LCH conducted additional numerical simulations of the prototype situation. The present case study allows thus to compare the wave characteristic derived from two classical engineering design tools: physical and numerical modeling. Furthermore, the latter results compared with the in-situ measurements, in order to assess the reliability of the models. Herein, the efficiency of the flap is thus not in the focus, but the comparison of the two modeling tools with the in-situ tests.

Table 1. Overview of the test program and conditions,

with PM = physical model, NM = numerical model, US =

upstream, and DS = downstream, all dimensions in prototype

values.

Test 1 2 3 4

Direction US DS US DS

Stern flap No No Yes Yes

V R [m/s] 0.35 (PM) 0.98 (in0.35 (PM) 0.98 (in0.98 (insitu, NM) 0.98 (insitu, NM) situ, NM) situ, NM)

V a convoy 4.55 (PM) 5.29 (in4.55 (PM) 5.20 (in

[m/s] 3.27 (insitu, NM) 3.35 (insitu, NM) situ, NM) situ, NM)

V r convoy 4.90 (PM) 4.25 (in4.90 (PM) 4.33 (in

[m/s] 4.31 (insitu, NM) 4.22 (insitu, NM) situ, NM) situ, NM)

D [m] 7.0 (PM) 8.7 (in7.0 (PM) 9.0 (in8.8 (insitu) 8.8 (insitu) situ, NM) 8.3 (NM) situ, NM) 8.8 (NM) f a 3.4 (PM) 82.2 (in3.4 (PM) 80.9 (in

[Hz] 129.3 (insitu) 132.6 (insitu) situ) 8.3 (NM) situ) 8.3 (NM) 8.2 (NM) 7.1 (NM)

2 WAVE MEASUREMENTS

2.1 General

Several tests were conducted with the different model types, of which only representative and, as far as possible, similar cases are selected for the herein given comparison, with the conditions as listed in Table 1. As the downstream journey is not critical for wave generation and the barge was empty for all journeys, downstream scenarios were ignored in the physical model tests and are thus missing in the herein presented comparison. The numerical model was set-up according to in-situ conditions (prototype scale), with the boundary conditions of the latter.

The following notation is use in Table 1: V_R = measured Rhone River flow velocity, V_a = absolute convoy velocity with the ground as reference, V_r = relative convoy velocity with Rhone River flow as reference, D = Distance between considered wave gauge (UDS) and convoy axis, and f_a = acquisition frequency.

2.2 Physical modeling

Physical model tests were conducted at 1:30 scale, based on the Froude similitude (LCH 2009, Fig. 2).The hull of the pusher tug and the barge were both mod eled using polystyrene foam. The ship models were loaded to adjust the gravity center as well as inertia, and painted for smoothening the surface. The pusher tug was connected to the barge by two rods allowing for a movement along the vertical axis. The tests were conducted in a channel, 2.0 m wide and 47 m long, regulating the flow depth with a shutter gate at the channel end. The discharge was supplied by in-house pumps and measured using a Magnetic Inductive Dis charge meter. The convoy was fixed on a motor driven Figure 2. Wave pattern (a) in-situ, as well as in the (b) numerical and (c) physical model. trolley with velocity control and measurement, which pulled the latter along the channel.The static water levels as well as the wave profiles were locally measured using Ultrasonic Distance Sensors (UDS) installed across the channel at distances D = 7, 12 and 24 m to the streamwise axis of the convoy, again in prototype dimensions. The channel bottom cross-section was rectangular, i.e. not reproducing the effective Rhone River bathymetry. The water depth was fixed at a representative value of 6.0 m, and the discharge was constant 123 m^3/s, both in prototype values. 2.3 Numerical modeling The numerical model was set-up in Flow-3D, version 10 (Flow Science 2011, Fig. 2). Flow-3D numerically solves the continuity and momentum equations using finite-volume approximation. The flow region is subdivided into a mesh of fixed rectangular cells. Within each cell, local averages of all dependent variables are associated. They are located at the center of the cells except for velocities, which are located at cell faces (staggered grid arrangement). Curved obstacles, wall boundaries, floating objects, or other geometric features are embedded in the mesh by defining the fractional face areas and fractional volumes of the cells that are open to flow (FAVOR method). Most terms in the equations are evaluated using the current time level values of the local variables explicitly. The two equation k-ε model is used for turbulence closure. The single incompressible fluid with a free surface model was used with no-slip condition on any solid surface boundary, i.e. the river bottom, pusher tug, and barge. The 3D geometry of the pusher tug and barge were inserted as two stereo-lithography (STL) files created in CAD, approximating the surfaces by triangles. The local bathymetry of the Rhone River was imple mented as a sweep of the effective profile with a streamwise regular cross-section width of some 90 m. The pusher tug and the barge were modeled as two individual floating objects, however with similar lon gitudinal velocities. Their densities were adjusted to achieve a correct immersion, and the gravity center as well as inertia were computed and validated. Near the convoy, a regular meshed grid with cell lengths of 0.100 m vertically, 0.525 m in streamwise direction and 0.500 m transversally was applied, in prototype values. First, the stationary state (water flows in the Rhone River and convoy is immobile at the model end) was simulated during 150 s for each test. Then, a restart was launched with flowing water plus a mov ing convoy, travelling along a defined path but free to translate in the vertical direction (heave) as well as to rotate around the transverse axis (pitch). The con voy advanced along 100 m in order to generate waves reaching the banks. The waves were derived from sec tions taken parallel to the path at different distances to the convoy.

2.4 In-situ measurements

A reach of the Rhone River with a straight, regular and streamwise constant cross-section of some 90 m width was chosen for the in-situ tests (LCH 2011, Fig. 2). No confluences join the Rhone River along the reach, so that the discharge and the velocities are con sidered as constant. The Rhone River discharge was 390 m^3/s provided by the Federal Office for the Envi ronment (FOEN) measurement station at Chancy, the flow velocity was measured usingADV (OTT Nautilus C2000), and the maximum flow depth at the talweg was derived as 7.0 m from the bathymetry data. The abso lute velocity of the convoy and its path were measured using on-board GPS as well as based on the measure ments of the hereafter described LDS. Note that the discharge was smaller in the physical model, being conducted first to define the flap shape.

A 20 m long beam was suspended below the bridge

“Passerelle du Lignon”, and equipped with four UDS

(Baumer UNAM S14) to record the water surface (wave profiles) and a horizontal Laser Distance Sensor (LDS, Micro-Epsilon ILR) to derive the distance between the UDS and the convoy. The effective distance between the UDS and the convoy axis varied at every passage, as a precise path was difficult do drive. The convoy with flap circulated at about 4.5 m distance from the beam, and the convoy without flap at about 2.4 m. As several UDS were mounted along the beam, the UDS closest to the aimed distance of some 9 m was taken into consideration for the comparison with the laboratory tests (Table 1). 3 COMPARISON OF CHARACTERISTIC PARAMETERS 3.1 Visual observations Figure 2 shows the wave pattern as observed in-situ, in the numerical and the physical model. A visual comparison indicates that: • Three dominant wave packages occur: (1) a first generated at the bow of the barge, (2) a second initiated at the stern of the barge respectively at the bow of the pusher, and (3) a third at the stern of the pusher. The third is, however, close to the second, so they are treated as one package. Note that there is neither an elbow wave for the barge nor for the pusher as they are straight. • The dominant waves appear basically similar in all pictures. • The water surface is generally smoother in the models than in-situ, where small waves are visible. 3.2 Wave profiles Wave profiles as measured at distance D according to Table 1 are shown in Fig. 3. The ordinate gives the water level, relative to the static elevation before the passage, and the abscissa gives the time t. The aforementioned dominant wave packages (1) and (2) are recognizable; namely (1) generated by the bow of the barge at t = 5 to 10 s including one or two waves, and (2) produced at the stern of the latter and at the bow of the pusher, respectively, at t = 15 to 20 s including several waves. The second package contains more waves, being affected by the mentioned flap. If focusing on the wave profiles as indicated by the two models and comparing them with the in-situ measurements, a analogy is evident. The physical model (PM) tends to underestimate the first wave crest at t = 7 to 8 s, whereas the wave troughs are correctly represented. Between the first and the second package, waves below the static level occur. As for the second package, the physical model indicates only one single dominant wave. The amplitude of the latter is overestimated. After the second wave package, water levels below the static elevation occur. In general, the physical model shows smooth waves without local irregularities. The numerical model (NM) tends to significantly underestimate the amplitudes at distance D according

Figure 3. Wave profiles for Tests (a) 1, (b) 2, (c) 3, and (d)

4 (Table 1), from in-situ data, numerical (NM) and physical

(PM) model.

to Table 1 due to numeric diffusion. Nevertheless, the first wave package is reconcilable and located at the correct position, but the second is hardly visible. The average water levels between the two wave packages are close to the static elevation. At the end of the mea surement reach, the numerical model indicates almost a plain water surface without waves, slightly below the static level. Again, the wave surfaces are smooth and free of local irregularities.

3.3 Wave heights

Box plots summarizing the wave profiles over 30 s of acquisition time (similar to Fig. 3) are shown in Fig. 4. Of particular interest are the outliers, which represent the maximum and minimum wave heights.

The physical model gives less outliers but with exag gerated values, whereas the numerical model indicates outliers reaching up to some 50% of the maximum in situ values. The boxes (including 50% of all points) of the physical model are located below the static level, as discussed in the context of Fig. 3, while those of

the numerical model are close to it. The box height

of the numerical model is slightly smaller than that of

the in-situ tests, whereas that of the physical model is

significantly bigger. Figure 4. Comparison of wave heights for Tests no (a) 1, (b) 2, (c) 3, and (d) 4 (Table 1). The wave damping of Test 1 as a function of D is shown in Fig. 5, based on the maximum amplitude of the wave profiles measured over 30 seconds (Fig. 3a). The reduction of the maximum amplitude is similar in-situ and in the physical model, but much smaller for the numerical simulation. Muk-Pavic et al. (2006) report of realistic wave features exclusively close-by a ship hull. The differences are more pronounced if the amplitude closest to the convoy (at minimum value D) is considered as reference, and relating the others to that value. The wave-damping results (Fig. 5b) indicate that the remaining maximum amplitude closest to

Figure 5. Maximum wave amplitudes as a function of the

distance D, (a) in absolute terms, and (b) damping relative to

the point measured closest to the convoy.

the river banks is 66% of the reference in-situ, around

59% in the physical model, and only 25% in the numer

ical model. The transversal damping of the maximum

amplitude is thus similarly represented by the physical

model, but overestimated by the numerical model.

3.4 Characteristic wave frequencies

The power spectral density of the observed waves is

estimated based on Welch’s averaged modified per

odogram method (Welch 1968). The resulting wave

frequencies f of the different measurements were thereafter corrected (subscript C) to consider the

slightly varying V_r, as the UDS were locally fixed,

while the convoy was travelling, so that

using “+” for journeys with the flow, and “–” against

it. As the minimum (subscript m) wave frequency

depends on the relative velocity V_r of the convoy (New

man 1978, Douglas et al. 1985), the power spectra were

derived only for higher frequencies than the related f_m

given as Figure 6. Power spectral densities for Tests no (a) 1, (b) 2, (c) 3, and (d) 4 (Table 1). Here, g = acceleration due to gravity. Furthermore, the upper frequency limit results from the sampling frequency of the laboratory tests as 1.7 Hz. In Fig. 6 the power spectral densities Pxx of the wave profiles of Fig. 3 are compared.The general trend of the curves is similar; in particular, the numerical model shows a reasonable agreement. The numerical model and the in-situ measurements both indicate a dominant frequency at 0.4 Hz, similar to a typical wave period of 2.5 s representing the two dominant wave

packages. The physical model correctly indicates high

energies in low-frequency waves, but gives no explicit

value.The high frequencies are underestimated in both

models. The physical model gives reliable values up to

some 0.8 Hz, whereas exceeding frequencies indicate

mainly noise.

4 DISCUSSION

Before discussing some observations, it has to be noted

that the conditions of the physical model differed from

that of the numerical model and the in-situ tests. The wave profiles of the physical model were recorded at some 80% of the relative distance to the convoy as compared to the numerical model and the in-situ tests, and that the relative convoy velocity was some 15% higher (Table 1). The laboratory flume has a rectangular cross-section and an up-scaled width of 60 m, instead of 90 m in-situ. Finally, the model con voy was pulled while in-situ a propeller feeds the wake zone. The herein presented comparison is thus only indicative.

Regarding a comparison of the wave profiles and therefrom extracted key parameters, the following could be identified regarding the reliability of physical and numerical modeling:

- The physical model correctly reproduces the wave heights, but gives no clear indication on the domi nant frequency.
- The numerical model underestimates the wave heights, but correctly reproduces the dominant fre quency. Note, however, that the underestimation increases with distance D to the convoy. Waves close to the convoy are represented almost correctly, whereas a numeric diffusion occurs with distance.
- After the first wave package, the physical model indicates too low water levels being significantly

below the static reference. This may be a consequence of the relatively narrow channel of the physical model, as compared to in-situ.

• Near the end of the modeled reach, both models indicate water levels slightly below the static reference, ignoring wake waves generated by the propeller. This may be explained with the fact that the UDS are fixed at a location in-situ giving the static level before passage of the convoy, whereas the numerical model gives a section parallel to the convoy course for a fixed time.

• Both models ignore small waves.

5 CONCLUSIONS

It may be assumed that the physical model basically gives accurate results in terms of wave heights, if the boundary conditions are correctly implemented. Information regarding the wave frequency and small waves

Example of wave impact on a residential house

D. Wüthrich, M. Pfister & A.J. Schleiss

Laboratoire de Constructions Hydrauliques (LCH), EPFL, Switzerland

ABSTRACT: Waves impacting against structures can create damages and devastation. This topic regained interest after some recent catastrophic events and the present paper investigates the main phases of a wave impacting against a residential house commonly observed in areas subject to tsunami hazards. The project is

based on an experimental approach and both dry bed surges and wet bed bores were tested. Through some visual observations and high speed cameras, important run-up heights in the vertical direction were observed for all configurations; these were more intense for wet bed bores. For the largest waves a full overflow of the structure was observed and the presence of the structure provoked a change of regime and the upstream propagation of a bore.

1 INTRODUCTION

In the past hydrodynamic waves such as dam-break waves, impulse waves and tsunamis were considered extremely rare events and most design codes do not present guidelines in this domain. Some recent catas trophic events with high casualties and significant damages showed the importance of wave-resistant houses as a mean to save people lives. On coastal areas around the Indian Ocean and at the mountains close to dam sights, most of the buildings do not exceed three floors, resulting into low structures characterised by a rectangular shape. Some previous relevant stud ies on wave impact against free standing structures were conducted by Cross (1967), Ramsden (1996), Arnason et al. (2009) and Nuori et al. (2010); it is also worth mentioning the Japanese contribution of Asakura et al. (2000) and Okada et al. (2005). Investi gations on the behaviour of residential houses subject to wave impact were carried out by Thusyanthan and Madabhushi (2008), who tested a “tsunami resistant house” designed by Harvard design school in collabo ration with MIT and compared it with a typical house in Sri Lanka. Wilson et al. (2009) and Van de Lindt (2009) tested a scaled wooden residential house in the US with both windows open and closed. The effect of porosity was tested by Lukkunaprasit et al. (2009). Chen et al. (2012) simulated the effect of overtop ping waves and Shafiei Amraei et al. (2011) provided some visual observations of tsunami impact on some typical coastal structures. In the context of a broader research project investigating in a laboratory environ ment the structural loading during wave impact, the hydraulic behavior of the process is here described and discussed. When a building is hit by an incom ing wave, the flow is 3-Dimensional, highly turbulent and many phenomena appear in a short period of time. Some visual observations were carried out using highspeed cameras to provide a better understanding of the time development of these phenomena. Results showed some high splashes and important run-ups, provoking the overflow for the highest waves. 2 EXPERIMENTAL SET UP For the present study a vertical release technique was used to produce both dry bed surges and wet bed bores; the same set-up was previously discussed and used by Wüthrich et al. (2016); similar techniques were used by Chanson et al. (2003), Meile (2007) and Rossetto et al. (2011). A known volume of water was released from an upper reservoir into a lower basin using three independent pipes with an internal diameter of 309 mm. If a dam-break configuration is considered, different discharges resulted into various equivalent impoundment depths, leading to waves with different characteristics in terms of height and approaching velocities. The propagation of the wave took place in a smooth horizontal channel with a length of 14 m and a width of 1.4 m, where the wave was followed in terms of velocity and height, using a UVP (Ultrasonic Velocity Profiler) and US (Ultrasonic distance Sensor) respectively. The US probes were located at x = 2, 10.1, 12.1, 13.1, 13.35, 13.6 and 13.85 m from the beginning of the channel. The building was located at a distance of 14 m from the channel inlet, allowing sufficient time for the wave to fully develop and reach a quasisteady condition. The structure consisted of a 30 cm impervious aluminum cube with stiffness estimated to $k = 2.2 \cdot 10^8$ N/m. Being the gravity effects predominant in this process, a Froude similitude with a scaling ratio of 1:30 was used and the structure corresponds

Figure 1. Experimental set-up and propagating channel.

Figure 2. Channel with aluminum building (H = 0.3 m).

to a building height of 9 m and to a wave ranging from 4.5 to 7.5 m, which is consistent with real observations of tsunamis and buildings commonly found in coastal areas. Some scale effects are expected in the turbulent mixing process, where viscous effects are dominant and where a Reynolds similitude should be used. At the present time no systematic study on the scale effects on propagating bores was carried out (Docherty and Chanson 2012), however the viscous scaling effects were minimized using large Reynolds numbers ($Re > 10^5$).

For all scenarios the maximum wave height (h_{max}) was lower than the building height (H), however for the higher waves, some overflow was observed. For the channel, the blockage ratio was $\beta = B/H = 4.6$.

This value was not sufficient to fully avoid side effects, however in real life the effect of the walls could be jus tified with the presence of other buildings on the side. The front velocity (V) was derived from the position of the front in time (h > 0.01 m), measured with the US probes.

3 EXPERIMENTAL RESULTS

To obtain a complete insight of the process, both dry bed surges and wet bed bores with different wave heights and wavelengths were tested and analyzed. Longer waves better represent tsunamis, whereas impulse waves are characterized by lower periods and higher amplitudes. The tested scenarios are presented in Table 1 along with the main hydrodynamic properties of the produced waves. 3.1 Dry bed surges Three surges on dry horizontal bed were tested; these were characterized by a thin front followed by a constant increase in water depth, until the maximum value was reached, then a constant decrease was observed. No aeration was noted in the propagating bore. Some secondary turbulence was observed on the surface of the surge, resulting from the internal turbulence of the flow. The waves were successfully compared with the theory developed by Ritter (1892) for a dam break scenario and three equivalent impoundment depths were used: d 0 = 0.4 m (1 pipe), d 0 = 0.6 m (2 pipes) and d 0 = 0.7 m (3 pipes), leading to wave heights and velocities presented in Table 1 (Wüthrich et al. 2016). The smallest surge impacted against the structure with the lowest front velocity, producing a reduced amount of splashes. For this scenario, no overflow of the structure was observed. When the tip hit the building some small vertical run up was measured, whose maximum height was slightly higher than the building (Figure 3a). Due to the transmission of momentum from the wave to the structure, the run-up moved backward, falling on the coming wave and producing a steady roller on the upstream side with air entrainment and high turbulence.The presence of the obstacle deviated the flow to both sides. The increasing discharge accentuated the deviation until the sides of the channel started having an influence (Figure 3b). As a consequence the wave height upstream of the building increased, the velocity decreased, leading to an important reduction of the Froude number. When the flow became subcritical a wave started propagating upstream. Behind the building the detachment of Von Kàrman vortexes were clearly observed. For increasing impoundment depths the surges presented higher approaching velocities and higher vertical run-ups were observed, reaching some two times the height of the structure (Figure 3d,g). The run-up fell on top of the building; the latter was submerged by the coming wave. Similarly to the previous case, with the increasing discharge, the flow was deviated to the side, reducing the approaching velocity and producing a wave propagating in the upstream direction. The behavior of the highest surge was similar to the middle one, however the first seconds were characterized by a pulsating behavior, probably due to the interaction between the incoming wave and the reflection of the vertical run-up; the building was totally overflowed. During the decreasing phase of the wave, the water depth at building sight reduced and the behavior was similar to a quasi-steady flow around a squared obstacle.

Table 1. Characteristics of the experimental tests carried out. Initial bed Number Equivalent impoundment Maximum wave Approaching condition of pipes depth d_0 [m] height [cm] velocity [m/s]

Surge 1	Dry	1	0.4	13.3	2.3
Surge 2	Dry	2	0.6	17.8	3.1
Surge 3	Dry	3	0.7	18.6	3.5
Bore 1	Wet, h_0 = 3 cm	3	0.7	25.4	2.8
Bore 2	Wet, h_0 = 5 cm	3	0.7	26.9	2.7
Bore 3	Wet, h_0 = 5 cm	1	0.4	18.5	1.9

Figure 3. Time lapse for the three tested surges: (a)-(c) d_0 = 0.4 m, (d)-(f) d_0 = 0.6 m and (g)-(j) d_0 = 0.7 m.

3.2 Wet bed bores

Waves propagating on an initial still water depth are commonly called bores and they have a completely different behavior than surges.These represent successive waves that propagates on the wet bed remaining from previous waves. Visually the bore appeared as a trans

lating roller with high air entrainment, followed by a

relatively constant water level; this configuration is consistent with the theory of Stoker (1957). The bores had lower velocities compared to dry bed surges and a steeper front for all configurations (Wüthrich et al. 2016). During the impact against the structure all bores with the same impoundment depth (d_0), independently from the initial still water depth (h_0), showed a similar behavior. Three initial impoundment depths

Figure 4. Time lapse for the three testes bores: (a)-(c) $d_0 = 0.7$ m, $h_0 = 3$ cm (d)-(f) $d_0 = 0.7$ m, $h_0 = 5$ cm and (g)-(j)

$d_0 = 0.4$ m, $h_0 = 5$ cm.

were tested, but only two are here presented ($h_0 =$

3 cm, 5 cm). Compared to the dry bed surges, for

all bores higher vertical run-ups were observed (Fig

ure 4a), probably due to the greater steepness of the

front. For all configurations with three pipes (3P,

$d_0 = 0.7$ m), the building is fully overflowed and sub

merged. For the bore produced with only one pipe (1P,

$d_0 = 0.4$ m), splashes higher than the building height

were observed, however the structure was not over

flowed. Similarly to the sur-ges, the constriction due

to presence of the building resulted into a change in the

flow regime with waves propagating in the upstream

direction. Like the previous case, the decreasing part of

the wave was similar to a steady flow around a squared

obstacle.

4 DISCUSSION AND CONCLUSIONS

Following the failure of a dam, a landslide into a lake or an offshore earthquake, a dam-break wave, an impulse wave or a tsunami are produced. Buildings located in the trajectory might be hit by this pro-pagating waves, leading to damages and devastation. The purpose of this study was to describe the main phenomena appearing during the impact of waves against an impervious 30×30 cm aluminum cube, reproducing a residential house of 9×9 m if a Froude scaling ratio of 1:30 is assumed. The project is based on an experimental approach and waves were produced using a vertical release technique. Given their different behavior, both dry bed surges and wet bed bores were tested. Highspeed cameras were used to gain a better insight and provide better support. Vi-sual observations proved that overall surges and bores with the same equivalent impoundment depths presented similar behaviors. For all scenarios the wave impact presented high splashes and some turbulent air entrainment on the upstream side of the building. In general bores showed higher splashes, probably due to their steeper front. The presence of the building provoked a constriction, with a decrease in flow velocity and an increase of the water level. The combination of these effects lead to a change in the flow regime and a propagation of a bore in the upstream direction. For all waves produced using three pipes ($d_0 = 0.7$ m) an overflow was observed and the structure was fully submerged. For the smallest waves the structure was not overflowed and the roof could be used as a vertical shelter in case of tsunami alert.As expected from literature, the visual observations proved a substantial difference in behav ior between bores and surges. The same difference was observed during the building impact, however at this early stage it is premature to decide which type represents the more critical scenario. Further investi gations are necessary and the measurements of impact forces essential to decide which type should be used to design wave-resistant houses. Furthermore, these findings only apply to impervious structures; the pres ence of opening such as windows and doors could completely change the behavior of the flow.

ACKNOWLEDGMENT

The support of the Swiss National Science Foundation (SNSF), grant 200021_149112/1 is acknowledged.

NOTATION

B channel width [m]

d_0 initial equivalent impoundment depth [m]

H building height and width [m]

h wave height [m]

h_0 initial water depth in the channel [m]

h_{max} maximum wave height [m]

k building stiffness [N/m]

t time [s]

x longitudinal direction of the channel [m]

β blockage ratio

Arnason H., Petroff C., and Yeh H. 2009. Tsunami bore impingement onto a vertical column. Journal of Disaster Research, 4(6):391-403.

Asakura R., Iwase K., Ikeya T., Takao M., Kaneto T., Fujii N.,

and Omori M. 2000. An experimental study on wave force acting on on-shore structures due to overflowing tsunamis.

Proceedings of Coastal Engineering JSCE, 47:911–915.

Chanson H., Aoki S., and Maruyama M. 2002. Unsteady

air bubble entrainment and detrainment at a plunging

breaker: dominant time scales and similarity of water level

variations. Coastal Engineering, 46(2):139–157. Chen X., Hofland B., Altomare C., Suzuki T., and Uijttewaal W. 2015. Forces on a vertical wall on a dike crest due to overtopping flow. Coastal Engineering, 95:94–104. Cross R. 1967. Tsunami surge forces. Journal of the Waterways and Harbors Division, 93(4):201–231. Docherty N.J and Chanson H. 2012. Physical Modeling of Unsteady Turbulence in Breaking Tidal Bores. Journal of Hydraulic Engineering, 138(5):412–419 Lukkunaprasit P., Ruangrassamee A., and Thanasisathit N. 2009. Tsunami loading on buildings with openings. Science of Tsunami Hazards, 28(5):303. Meile T. 2007. Influence of macro-roughness of walls on steady and unsteady flow in a channel. Ph.D. Thesis, EPFL-LCH, Lausanne Switzerland, 414 pages. Nouri Y., Nistor I., Palermo D., and Cornett A. 2010. Experimental investigation of tsunami impact on free standing structures. Coastal Engineering Journal, 52(01):43–70. Okada T., Sugano T., Ishikawa T., Ohgi T., Takai S., and Hamabe C. 2005. Structural design method of buildings for tsunami resistance. BuildingTechnology Research Institute, Building Centre for Japan, 15. Ramsden J. 1996. Forces on a vertical wall due to long waves, bores, and dry-bed surges. Journal of waterway, port, coastal, and ocean engineering, 122(3):134–141. Ritter A. 1892. Die Fortpflanzung der Wasserwellen. Zeitschrift Verein Deutscher Ingenieure, 36, 33, 947–954 Rossetto T., Allsop W., Charvet I., and Robinson D. 2011. Physical modelling of tsunami using a new pneumatic wave generator. Coastal Engineering, 58(6):517–527. Shafiei Amraei SR., Melville B.W., Shamseldin A.Y., Burridge B. and Hogan A. 2011. Tsunami impact on Coastal Structures. 34th IAHR World Congress – Balance und Uncertainty, Brisbane, Australia, 26 June–1 July Stoker J.J. 1957.WaterWaves:The MathematicalTheory with Applications. Intersciences, 567 pages Thusyanthan N. I., & Gopal Madabhushi S. P. 2008. Tsunami wave loading on coastal houses: a model approach. Proceedings of the Institution of Civil Engineers-Civil Engineering, 161(2):77–86. Van de Lindt J., Graettinger A., Gupta R., Skaggs T., Pryor S., and Fridley K. 2007. Performance of

wood-frame structures during Hurricane Katrina. Journal of Performance of Constructed Facilities, 21(2):108–116.
Wilson J., Gupta R., Van de Lindt J., Clauson M., and Garcia R. 2009. Behavior of a one-sixth scale woodframed residential structure under wave loading. Journal of Performance of Constructed Facilities, 23(5):336–345.
Wüthrich D., Nistor I., Pfister M. and Schleiss A.J. 2016. Experimental generation of tsunami-like waves. Proceedings of Coastal Structures. Boston, MA, USA. 9–11 September (under publication).

Baloch, M.A., Ames, D.P. & Tanik, A. 2015. Hydrologic impacts of climate and land use change on Namnam Stream in Koycegiz Watershed, Turkey. Int. J. Environ. Sci. Technol. 12: 1481-1494.

De Silva, M.M.G.T., Weerakoon S.B. & Herath, S. 2014. Modeling of Event Based and Continuous Flow Hydro graphs with HEC-HMS: Case Study in the Kelani River Basin, Sri Lanka, Journal of Hydrologic Engineering 19(4): 800-806.

Institute of Hydrology. 1980. Calculation of the Base Flow Index. Report No 108 Low Flow Estimation in the United Kingdom.

Joo, J., Kjeldsen,T., Kim, H.J. & Lee, H. 2014.A Comparison of Two Event-based Flood models (ReFH-rainfall Runoff Model and HEC-HMS) at Two Korean Catchments, Bukil and Jeungpyeong. KSCE Journal of Civil Engineering 18(1): 330-343.

Josephine, V.S., Mudgal, B.V. & Thampi, S.B. 2014. Appli cability of Doppler weather radar based rainfall data for runoff estimation in Indian watersheds-A case study of Chennai basin. Sadhana 39(4): 989-997.

Piman , T. & Babel M.S. 2013. Prediction of Rainfall-Runoff in an Ungauged Basin: Case Study in the Mountain ous Region of Northern Thailand. Journal of Hydrologic

Engineering 18(2): 285–294.

Razavi, T. & Coulibaly, P. 2013. Streamflow Prediction in

Ungauged Basins: Rewiev of Regionalization Methods.

Journal of Hydrologic Engineering 18(8): 958–975.

Reshma, T. Reddy, K.V. & Pratap, D. 2013. Simulation

of Event Based Runoff Using HEC-HMS Model for an Experimental Watershed. International Journal of Hydraulic Engineering 2(2): 28–33. Reshma, T., Reddy, K.V., Pratap, D., Ahmedi, M. & Agilan V. 2015. Optimization of Calibration Parameters for an Event Based Watershed Model Using Genetic Algorithm. Water Resources Management 29: 4589–4606. US Army Corps of Engineers Hydrologic Engineering Center (USACE). 2013. Hydrologic Modeling System HEC-HMS User's Manual. (http://www.hec.usace.army.mil/software/hec-hms/documentation/HEC-HMS_Users_Manual_4.0.pdf). Yanmaz, M. 2006. Applied Water Resources Engineering. Ankara: METU PRESS. Yener, M.K., S¸orman, A.Ü., S¸orman, A.A, S¸ensoy, A. & Gezgin, T. 2007. Modeling Studies with HEC-HMS and Runoff Scenarios inYuvacık Basin, Turkiye. International Congress on River Basin Management, Antalya, 22–24 March 2007. Yılmaz, A.G., Imteaz, M.A. & Ogwuda, O. 2012. Accuracy of HEC-HMS and LBRM Models in Simulating Snow Runoff in Upper Euphrates Basin. Journal of Hydrologic Engineering 17(2): 342–347. Zhang, H.L., Wang, Y.J., Wang, Y.Q., Li, D.X. & Wang X.K. 2013. The effect of watershed scale on HEC-HMS calibrated parameters: a case study in the Clear Creek watershed in Iowa, US. Hydrol. Earth Syst. Sci. 17: 2735–2745.

Using ANN and ANFIS Models for simulating and predicting Groundwater Level Fluctuations in the Miandarband Plain, Iran

M. Zare & M. Koch

Dept. of Geotechnology and Geohydraulics, University of Kassel, Kassel, Germany

ABSTRACT: The Miandarband plain is one of the most fertile plains of the Kermanshah province, Iran. The major water supply for agriculture is groundwater. In this regard, simulation and prediction of groundwater level (GL) fluctuations plays an important role for effective water resources management. GL-changes are complex to model, as they depend on many nonlinear and uncertain factors, thus, selecting suitable numerical or stochastic models that could simulate the nonlinearity and complex patterns is of great importance. Artificial Neural Networks (ANN) and/or fuzzy logic models are one family of models that have proven to be very useful to that regard. In this study, after data completion, using a novel multiple linear regression approach the Feed Forward Neural Network (FFNN) model with one hidden layer whose perceptrons have been optimized in the -training phase with three methods (Levenberg-Marquardt, Bayesian Regularization and Scaled Conjugate Gradient) and the Adaptive Network-Fuzzy Inference System (ANFIS) have been applied and evaluated for GLfluctuations simulation and prediction in the Miandarband plain, Iran. The results show that both model approaches can be used with acceptable accuracy, wherefore theANFIS-model performs better than the three FFNN-model variants.

In fact, the values of R^2 and RMSE for ANFIS are 0.97 and 0.48, respectively, in the training phase and 0.96 and 0.52 in the testing phase.

1 INTRODUCTION

Changing hydrological conditions occurring, for example, in the wake of future climate change (IPCC, 2007) by alterations of temperatures and precipita tion, will have detrimental effects on the surface and groundwater resources in many areas of the world

(Koch, 2008). This holds particularly for regions and countries which are already nowadays affected by water scarcity, such as the Middle Eastern region, including Iran. There, responding also to the needs of a strongly increasing population, rising water with drawals have already caused drastic changes in the surface flow regimes and severe drops of groundwater levels in many watersheds of that country (Zare and Koch, 2014a).

Because of Iran’s location in an arid and semi-arid climate region, groundwater is a major water sup ply for domestic, agricultural, and industrial users (Taheri and Zare, 2011). In this regard, simulation and prediction of groundwater level (GL) fluctua tions plays important role for effective water resources management, or in other words, accurate groundwa ter level simulations will help water authorities to better plan effective groundwater utilization (Todd and Mays, 2005). GL-changes in a drafted aquifer are not only affected by groundwater pumping, but more importantly by other hydrological variables such as rainfall, evaporation and recharge through the unsaturated zone, all of which are highly nonlinear, stochastic, and complex processes (Srivastav et al., 2007).Therefore, finding suitable models that can simulate GL-fluctuation changes is an ongoing challenge for hydrogeologists. Nowadays, data-driven computing tools such as artificial neural networks (ANNs) and fuzzy logic have been used in various fields of science and technology for modeling purposes (Brion et al., 2002).These techniques, also called artificial intelligence methods, are based on ideas how information is processed in biological systems. One of the advantages of such "soft" computing methods in system modeling is getting accurate results without having well-defined nonlinear physical relations between variables (Nayak et al., 2004). In recent years, ANN and fuzzy methods have been widely employed in groundwater studies (Umamaheswari and Kalamani, 2014). Azhar and Watanabe (2007) used ANN and ANFIS models for predicting daily GLfluctuation in Saitama city, Japan, in two groundwater wells. The results showed that GL could be predicted by these soft computing models with high accuracy. Shirmohammadi et al. (2013) tested and evaluated several data-driven techniques such as ANN, ANFIS and time series models for GLpredictions in the Mashhad plain, Khorasan Razavi province, Iran. The authors showed that ANFIS has a better performance than other models in predicting GLs one and two months ahead. Emamgholizadeh et al. (2014) studied the potential of ANN and ANFIS

Figure 1. The Miandarband plain in west Iran.

for the forecasting of GLs in the Bastam plain in Iran and indicated that these tools could predict the GLs accurately, wherefore the ANFIS-model, resulting in an $R^2 = 0.96$, performed better than the ANN-model with an R^2 of only 0.83. Maiti and Tiwari (2014) applied bothANN andANFIS model in drought-prone area of Dindigul, Southern India. They also achieved better GL-prediction results for theANFIS-model than for both the BNN.SCG and ANN.SCG versions of the ANN-model. Outside of Iran, Sirhan and Koch (2013) applied classical ANN to predict dynamic GL's in the Gaza coastal aquifer.

In this study, the Feed Forward Neural Network

(FFNN)ANNmodel with three training methods (Levenberg-Marquardt, Bayesian Regularization and Scaled Conjugate Gradient) and theANFIS-model will be used for GL-fluctuations simulations and predic tion in the Miandarband floodplain in western Iran which has been the focus of several recent studies of the authors (Zare and Koch, 2014a, Bishop, 1995).

2 MATERIALS AND METHODS

2.1 Study area

The Miandarband plain is located in western Iran, near the city of Kermanshah, between latitudes $34^{\circ} 24' 03''$ – $34^{\circ} 40' 56''$ and longitudes $46^{\circ} 44' 26''$ –$47^{\circ} 11' 00''$ .This region is geographically limited in the North by the Gharal and Baluch mountains and in the South by the Gharsu river and has a surface area of about 300 km^2 (see Figure 1). Surface water in the study area occurs in the form of springs and stream flow, with the major river being the Razavar river (Zare and Koch, 2014a).

2.2 Data description

For simulating and predicting the GL-fluctuations in the Miandaraband plain, precipitation and piezometric head data, recorded on a monthly base at 33 wells – seeTable 1 during the time period October 1991-June 2013 (261 months), are used. The locations of these wells with their Thiessen polygons are illustrated in

Figure 2. Table 1. List of the 33 wells and the areas of their Theissen polygons. ID Location A (ha) ID Location A(ha) 1 Biabr 1008 18 Tekea 750 2 Pirmazd 1005 19 Siachagha 1016 3 Baktashabad 955 20 Valiabad 1127 4 Varela 638 21 Soltankuh 1319 5 Sarvaran 552 22 Amirabad 734 6 Dahbagh 894 23 Nafte & Rika 547 7 Mah. Abad 1178 24 Meymunbaz 512 8 Goharchagha 867 25 Berimavand 736 9 Ahmadabad 911 26 Yavari 1211 10 Laelabad 859 27 BelaKabud 1337 11 Hashilan 1123 28 Hojatabad 1531 12 Jologir 1082 29 Nezamabad 1295 13 Kolakabud 616 30 Docheshme 579 14 Khoshinan 971 31 Ghazanchi 1406 15 Dehkoor 705 32 Nazarabad 338 16 Tasolejan 1293 33 Pirhayati 781 17 Tapeafshar 914 Figure 2. Well locations with Thiessen polygons. Figure 3 shows the monthly precipitation data recorded during the same time period at the meteorological station Kermanshah. The average annual precipitation which in the regions occurs in the form of snow and rain in the 1991–2013 time period, amounts to 410.5 mm which is more than the average annual precipitation of ~250 mm for Iran. 2.3 Completion of missing groundwater level values Complete raw GL-data for the period 1991–2013 were only available for 26 out of 33 wells. For the remaining 7 wells (see Table 2), the missing monthly values for

Table 2. # of missing data for wells with data gaps.

Well ID	8	12	16	19	21	24	29
# of missing data	4	4	34	3	6	38	24

Figure 3. Time series of Kermanshah monthly precipitation.

some wells – whose numbers range between 4 and 35 out of a total of 261 observations – were estimated by inverse distance weighting (IDW) interpolation of GL data from surrounding wells, wherefore consistency was checked by comparing the estimated missing value with those from the previous and next month. For wells with more than 6 monthly missing data (3 wells), Mul tiple Linear Regression (MLR) was employed. In this case, the correlation coefficients r between the incom plete GL-time series of the well under question and the

other 32 wells were firstly calculated and then the GL

data of the two wells (x_{i1} , x_{i2}) with the highest r were

selected as predictors in the MLR-model to predict

the missing GL-values y_i . The MLR-equation can be

written as:

or in matrix notation

where X is the N × 3 predictor matrix – with N = 261,

the total number of GL-time series values-, β is the

unknown regressor-vector, and ε is the error term,

accounting the unexplained noise in the data and/or

model errors (Zare and Koch, 2014b).

Once the linear model (2) is solved by a

least-squares approach, the accuracy of the MLR

prediction/interpolations evaluated by the coefficient

of determination R^2 and the Root Mean Square Error

(RMSE), defined by

are used, where y_i , $\bar{y}$ and $\hat{y}_i$ are observed, average of

observed and the MLR-calculated GL-datum in month

i, respectively. Figure 4. Left panels: Correlation coefficients between the GL-data of the three wells (top to bottom) with missing values and the other 32 wells, Right panels: MLR predictor surfaces for these three wells. Table 3. MLR modeling results. ID of ID of well with two most MLR RMSE miss. data corr. wells equation R^2 (m) 16 10, 22 y_{16} = 46565 – 0.91 1.68 35.2 x_{10} – 35.6x_{22} 24 17, 18 y_{24} = 6030.7 – 0.79 0.93 4 x_{17} – 3.8 x_{18} 29 20, 22 y_{29} =-13073 + 0.87 1.27 10.5x_{20} + 11x_{22} Figure 4 shows the r-values calculated between each of the three wells with missing values and the other 32 wells. For the two wells with the highest r values, the corresponding MLR-equation (1) is set and solved.The response surfaces for the MLR-predictions with the two predictor wells are also shown in the figure, whereas Table 3 lists the corresponding MLRequations, together with the statistical indicators of the regressions. After data completion, a weighted average (using the 33 Thiessen polygon areas as weights) hydrograph of the GL-fluctuations has been generated which is illustrated in Figure 5. 2.4 Multilayer feed forward neural network FFNN The basic concept of an artificial neural network (ANN) is derived from an analogy with the biological nervous system of the human brain and how the latter processes information through its millions of neurons interconnected to each other by synapses. Borrowing this analogy, an ANN is a massively parallel system composed of many processing elements (neurons),

Figure 5. Average GL-time series in the Miandarband plain.

where the synapses are actually variable weights, specifying the connections between individual neurons and which are adjusted, i.e. may be shut on or off during the trainingor learning phase of the ANN, similar to what happens in the biological brain (Sirhan and Koch, 2013).

Multilayer FFNN is one of the most popular and most widely used ANN-models. It is also a biologically inspired classification algorithm and consists of a number of simple neuron-like processing units, organized in layers. Every unit in a layer is connected with all the units in the previous layer (Figure 6) by so-called weights w. The latter encode the knowledge about the network and are estimated during the training process, discussed below. Data enters at the inputs and passes through the network, layer by layer, until it arrives at the outputs. During normal operation, that is when it acts as a classifier, there is no feedback between lay ers. This is why they are called feed-forward neural networks.

When the network-weights and -biases are initial ized, the network is ready for training. The multilayer FFNN can be trained for nonlinear and complex pat terns, such as the monthly GL's in the study region, as has been done likewise in the Gaza aquifer study of Sirhan and Koch (2013). The training consists math ematically essentially of the adaptive computation (back-projection) of the weights between the vari ous inputand output-units, by a local, or better, a global optimization method, such that some (squared) error (objective) function $|E(w)|^2$ between observed andANN-predicted output is minimized.The selection of the most appropriate optimization/minimization technique has been an ongoing challenge in ANN research (Markovic and Koch, 2005, Heaton, 2005). Three training methods, Levenberg-Marquardt (LM), Bayesian Regularization (BR) and Scaled Conjugate Gradient (SCG), will be used in the present applica tion.

The LM-algorithm is based on the generally well known Levenberg-Marquardt optimization technique

(Levenberg, 1944, Zare and Koch, 2016). It is essen

tially a combination of the steepest descent and the

Gauss–Newton algorithm. The LM-technique uses the

following equations to update the neuronal weights w: Figure 6. Architecture of FFNN with one hidden layer. where J is the Jacobian matrix,μ is a damping constant which controls the change of the method from steepest descent (μ→∞) to Gauss-Newton (μ→ 0), and I is an identity matrix. μ is varied constantly during the iterative minimization process, to gear the updated solution vector w ji (t + 1) monotonically towards the minimum of the objective function. In the second back-projection method, Bayesian Regularization (BR) (Beale et al., 2015, Markovic and Koch, 2015), the weights and the bias values are basically also updated as in the Levenberg-Marquardt procedure. However, whereas in the LM-method the damping or regularization parameter μ is adapted iteratively, in the BR-method it is computed based on some a priori (Bayesian) information on the unknown variances, assuming that the weights and biases of the network are random variables with specified distributions. In terms of the optimization (minimization) process, this is equivalent to a combined minimization of the squared errors |E(w)| 2 and the weights w By this procedure, whatever the size of the network, the objective error function will not be over-fitted, as it is partly penalized by the subjective a priori information on the unknown solution (Daliakopoulos et al., 2005). The third optimization method employed, SCG, belongs to a special class of conjugate gradient methods which requires no iterative line search and is fully automated, so that none of the unknown parameters depend on external user choices Being a Conjugate Gradient Method which has superlinear convergence for most problems, and by avoiding the time consuming line-search per learning iteration, the stepsize scaling mechanism of the SCG makes this algorithm faster than other second order optimization algorithms which are usually based on the full or partly approximation of the Hessian-matrix of the objective function (Møller, 1993).

Figure 7. Architecture of ANFIS.

2.5 Adaptive neuro fuzzy inference system/ANFIS

ANFIS are a class of adaptive neural networks that

are functionally equivalent to fuzzy inference sys tems. Jang (1993) combined both Fuzzy Logic and network-based model like ANN to produce a pow erful processing tool named ANFIS. This approach has some advantages over the classical ANN, such as the capability of a large amount of data storage, dynamic and nonlinear systems modeling, easy to use, high-speed model development, reducing of comput ing time, while still exhibiting increased estimation and prediction accuracy. Combining at the same time the benefits and capabilities of neural network struc ture methods and Fuzzy Logic, ANFIS uses a hybrid approach of the classical gradient descent procedure and systematic back-propagation tries to avoid the "trap" of the error function in a local minimum. (Tah masebi and Hezarkhani, 2012). The adaptive network based on the Sugeno fuzzy inference model provides a deterministic system of output equations, and is so a useful approach for parameter estimation (Takagi and Sugeno, 1985). The ANFIS approach sketched in Figure 7 has five layers (Jang, 1993).

The first layer, called the input layer $O_{1,i}$, is the output of the i-th node of the layer 1

Where $x_{1,2}$ is the input node i and A_i (or B_i) is a lin guistic label associated with this node.Therefore $O_{1,i}$ is the membership grade of a fuzzy set (A_1 , A_2 , B_1 , B_2

).

The second layer is the rule node with AND and/or

OR operators. The output $O_{2,i}$ is the product of all the

incoming signals.

The third layer outputs ($O_{3,i}$) are called normalized

firing strengths. The fourth layer contains the so-called consequent nodes which are standard perceptrons. Every node in this layer is an adaptive node with a node function: Where p_i, q_i, r_i is the parameter set of this node; called also consequent parameters. The fifth layer is called the output layer where the overall output is computed as the summation of all incoming signals. 2.6 Training and testing of groundwater FFNNand ANFIS-prediction models Both the FFNNand ANFIS-models are trained on the 33 monthly observed (and corrected for missing data) well-groundwater level (GL) and precipitation data series. The following input-output prediction model for GL at time t, GL_t is used: i.e. it is assumed that the groundwater levels in one particular month t, GL_t depend on up to t-i previous months'groundwater levels and precipitation P_{t-j}. Although Eq.(12) is basically just a simple transfer model, it has some physical basis, as groundwater data usually exhibits (1) some persistence, as illustrated in Figure 5, and (2) the rainfall needs some time to recharge the groundwater aquifer (Pedro-Monzonís et al., 2015, Hybel et al., 2015). For determining the maximum i and j, the autocorrelation of the GL-time series and the cross correlation of the GLwith the precipitation time series P are calculated, respectively. The continuous autocorrelation function R_{xx} of a time series x is defined as (Taghizadeh, 2000): where T is the period of observation, and l is the lag time. For discrete data, Eq. (13) is replaced by its discrete homologue which for x = GL, reads where N (=261) is the number of observing months. The cross-correlation function R_{xy} between two time series x and y is defined similarly to Eq. (13), so that the discrete cross correlation between GL and P at lag l is then written as:

Figure 8. Autocorrelation and cross correlation function.

Figure 8 shows that GL_t is mostly correlated with

$GL_{t-1,t-2,t-3}$ and $P_{t-1,t-2,t-3,t-4}$. Therefore, the final inputoutput FFNN and ANFIS model (Eq. 12) can be written as:

For the training and testing of the models, the observed GL and P data series have been randomly divided into two sets, wherefore the length of the train ing data set is 70% and test (prediction) data makes up the remaining 30%. Moreover, in the training phase of the FFNN, only 70% of the original training data is used for training per se, 15% for validation and 15% for testing during training in this phase, i.e. the latter forming an independent test data which dif fers from the 30% applied for simulation (prediction) with the trained network. By this approach, which dif fers slightly from the usual training, validation and test approach, the iterative, epochal update of the FFNN-model could be improved.

After determining the inputs and output of the FFNN-model, number of layers, number of neurons in the hidden layer, the transfer function should be deter mined. As Figure 6 shows, the selected FFNN has an input, one hidden and an output layer. The transfer functions are selected by trial and error method based on best training performance. By doing so, the tan sigmoid (tansig) function was found to be best working between input and hidden layer and the linear (purelin) transfer function between hidden and output layer.

Determining the number of neurons in the hidden

layer (s) is one of the most important and difficult

tasks in FFNN-studies. Using too few or too many

neurons in the hidden layers will result in underor

overfitting of the model, respectively. In addition, large

number of neurons in the hidden layer will increase the

computational training time. There are many rules of Figure 9. RMSE for each number of neurons in hidden layer. thumb for determining the correct number of neurons in the hidden layers, such as a) the number of hidden neurons should be in the range between those in the input and output layers (between 1 and 7); b) it should be 2/3 of those in the input layer, plus those in the output layer which is equal to 6 neurons; c) it should be less than twice the input layer size (Heaton, 2005). Based on these rules, FFNN-models with up to 14 neurons in the hidden layer have been tested and the corresponding RMSE of the trained model measured. As Figure 9 shows, the optimal number of neurons in the hidden layer is 10. Once the optimal FFNN-structure, i.e. a 7-10-1percepteron model, has been determined, the model is trained and simulated by LM-, BRand SCG-training methods, discussed in the previous section. The ANFIS-modeling approach requires a further division of the input/output data into rule patches. In standardANFIS, either grid partitioning or the subtractive clustering method is applied.The problem with the former is that as the number of inputs increases, the number of rules rises rapidly. In other words the fuzzy rules increase exponentially, such that if, for example m membership functions (MF) are determined for each of n input variables, the number of fuzzy rules will be m n (Vaidhehi, 2014). This, of course, means a tremendous load on the computer processor and memory requirements. For remedy, this large number of fuzzy rules is reduced by integrating ANFIS with Fuzzy Clustering Method (FCM). The FCM is used to systematically create the fuzzy MFs and the rule base for ANFIS (Abdulshahed et al., 2015). 3 RESULTS AND DISCUSSION For the statistical analysis of the results of both the various FFNN-variants and ANFIS models, R 2 and RMSE parameters have been computed for both the training and the test (prediction) phases (see Table 4 and Figures 10 to 13). Table 4 shows that the minimum RMSE and the maximum R^2 in training phase with 0.48 and 0.967, respectively, are obtained with the ANFIS-model. The latter exhibits also the best results in the test phase, with corresponding values of 0.52 and 0.962. The reason for the superiority of ANFIS over the three FFNN-models may be related to the combination of

Table 4. Training and test results of models.

	Training		Test	
Method	RMSE	R^2	RMSE	R^2
FFNN-LM	0.61	0.965	0.88	0.920
FFNN-BR	0.68	0.939	0.98	0.875
FFNN-SCG	0.71	0.927	0.69	0.930
ANFIS	0.48	0.967	0.52	0.962

Figure 10. Regression plots of ANFIS.

Figure 11. FFNN-LM results.

Figure 12. FFNN-BR results.

fuzzification of the input through membership functions with the network-based algorithm and the use of the hybrid (back-propagation tries and gradient descent) optimization method. Figure 10 shows the regression plots between ANFIS calculated Gls and the observed data.

The Figures 11 to 14 reveal that the absolute maximum errors for one month are only 2.25 m and 2.03 m in training and test phases, respectively, for theANFIS model, compared with 2.20 m, 2.92 m for FFNN-LM; 6.5 m, 5.47 m for FFNN-BR; and 3.65 m, 3.45 m for FFNN-SCGThese results clearly show thatANFIS has better performance than ANN (FFNN) for simulating and predicting the GL-fluctuations in the study region. Figure 13. FFNN-SCG results. Figure 14. ANFIS results. Figure 15. Random errors distribution of ANFIS. All data driven methods are based on the idea that the random errors are drawn from a normal distribution and ANFIS is not an exemption. Figure 15 illustrates that the a posteriori computed errors of the GLs follow indeed such a normal distribution. 4 CONCLUSIONS In this study, firstly a MLR-based method for estimating missing data values in 33 groundwater level (GL) well series in the Miandarband plain (Iran) was developed. Using three back-projection variants of FFNN as well asANFIS, a dynamic model for the GL's as a function of past GL's and precipitation was then set up. The statistical results, evaluated by RMSE and R^2, indicate that ANFIS outperforms all three FFNN model variants with values of RMSE and R^2 of 0.48 m and 0.97, respectively, in the training phase and of 0.52 and 0.96 in the testing (prediction) phase.These results illustrate that FFNN but more so, ANFIS can provide reliable conceptual models for groundwater level prediction under different water resources management scenarios and can so be valuable modeling tools for groundwater resources planning projects.

Abdulshahed,A. M., Longstaff,A. P., Fletcher, S. & MYERS, A. 2015. Thermal error modelling of machine tools based on ANFIS with fuzzy c-means clustering using a ther mal imaging camera. Applied Mathematical Modelling, 39, 1837-1852.

Azhar, K. A. & Watanabe, K. 2007. Daily groundwater forecasting using soft computing technique. Nature and Science, 5, 1-10.

Beale, M. H., Hagen, M. T. & Demuth, H. B. 2015. Neural Network Toolbox™Getting Started Guide, The MathWorks, Inc.

Bishop, C. M. 1995. Neural Networks for Pattern Recogni tion, Oxford University Press, Inc.

Brion, G. M., Neelakantan, T. R. & Lingireddy, S. 2002.
A neural-network-based classification scheme for sorting
sources and ages of fecal contamination in water. Water
Research, 36, 3765–3774.
Daliakopoulos, I. N., Coulibaly, P. & Tsanis, I. K. 2005.
Groundwater level forecasting using artificial neural net
works. Journal of Hydrology, 309, 229–240.
Emamgholizadeh, S., Moslemi, K. & Karami, G. 2014. Pre
dicting of the groundwater level of Bastam Plain (Iran) by
Artificial Neural Network (ANN) and Adaptive Neuro
Fuzzy Inference System (ANFIS). Water Resources Man
agement, 28, 5433–5446.
Heaton, J. 2005. Introduction to Neural Networks with Java,
Heaton Research Inc.
Hybel,A. M., Godskesen, B. & Rygaard, M. (2015) Selection
of spatial scale for assessing impacts of groundwater
based water supply on freshwater resources. Journal of
Environmental Management, 160, 90–97.
IPCC. 2007. Climate Change 2007: The Physical Science
Basis. Contribution of Working Group I to the Fourth
Assessment Report of the Intergovernmental Panel on Cli
mate Change [Solomon, S. et al. (eds.)] Cambridge, UK.,
Cambridge University Press.
Jang, J. S. R. 1993. ANFIS: Adaptive-Network-based Fuzzy
Inference Systems. IEEE Transactions on Systems, Man

and Cybernetics., 23, 665-685.

Koch, M. 2008. Challenges for future sustainable water resources management in the face of climate change. 1st NPRU Academic Conference. Nakhon Pathom University, Thailand.

Levenberg, K. 1944. A Method for the Solution of Cer tain Non-Linear Problems in Least Squares. Quarterly of Applied Mathematics, 2, 164-168.

Maiti, S. & Tiwari, R. K. 2014. A comparative study of artificial neural networks, Bayesian neural networks and adaptive neuro-fuzzy inference system in groundwa ter level prediction. Environmental Earth Sciences, 71, 3147-3160.

Markovic, D. & Koch, M. 2005. Wavelet and scaling analy sis of monthly precipitation extremes in Germany in the 20th century: Interannual to interdecadal oscillations and the North Atlantic Oscillation influence. Water Resources Research, 41.

Markovic, D. & Koch, M. 2015. Stream response to precipita tion variability: a spectral view based on analysis and mod elling of hydrological cycle components. Hydrological

Processes, 29, 1806-1816. Moller, M. F. 1993.A scaled conjugate gradient algorithm for fast supervised learning. Neural Networks, 6, 525-533. Nayak, P. C., Sudheer, K. P., Rangan, D. M. & Ramasastri, K. S. 2004. A neuro-fuzzy computing technique for modeling hydrological time series. Journal of Hydrology, 291, 52-66. Pedro, M. M., Solera, A., Ferrer, J., Eetrela, T. & Paredes, A. J. 2015. A review of water scarcity and drought indexes in water resources planning and management. Journal of Hydrology, 527, 482–493. Shirmohammadi, B., Vafakhah, M., Moosavi, V. & Moghaddamnia, A. 2013. Application of Several Data-Driven Techniques for Predicting Groundwater Level. Water Resources Management, 27, 419–432. Sirhan, H. & Koch, M. 2013. Prediction of Dynamic Groundwater Levels in the Gaza CoastalAquifer, South Palestine, using Artificial Neural Networks. Journal of Hydrocarbons, Mines and Environmental Research (JHMER). 4, 42–56. Srivatsav, R. K., Sudheer, K. P. & Chaubey, I. 2007. A simplified approach to quantifying predictive and parametric uncertainty in artificial neural network hydrologic models. Water Resources Research, 43. Taghizadeh, S. R. 2000. Part 3Discrete-Time Signals & Systems Case Studies. Digital Signal Processing. School of CommunicationsTechnology and Mathematical Sciences. University of North London, UK. Taheri, A. & Zare, M. 2011. Groundwater artificial recharge assessment in Kangavar Basin, a semi-arid region in the western part of Iran. African Journal of Agricultural Research, 6, 4370–4384. Tahmasebi, P. & Hezarkhani, A. 2012. A hybrid neural networks-fuzzy logic-genetic algorithm for grade estimation. Computers & Geosciences, 42, 18–27. Takagi, T. & Sugeno, M. 1985. Fuzzy Identification Systems and Its Applications to Modeling and Control. IEEE Transactions on Systems, Man and Cybernetics, SMC-15, 116–132. Todd, D. K. & Mays, L. W. 2005. Groundwater Hydrology, New York, John Wiley and Sons Inc. Umamaheswari, G. R. & Kalamani, D. (2014) Fuzzy logic model for prediction of groundwater level in Amaravathi river minor basin. International Journal of Mathematics trends and technology, 11, 46–50. Vaidhehi, V. 2014. The role of Dataset in training ANFIS system for course Advisor. International Journal Innovative Research in Advanced Engineering (IJIRAE), 1, 249–253. Zare, M. & Koch, M. 2014a. 3Dgroundwater flow modeling of the possible effects of the construction of an irrigation/drainage network on water-logging in the Miandarband plain, Iran. Basic Research Journal of Soil and Environmental Science, 2, 29–39. Zare, M. & Koch, M. 2014b. An Analysis of MLR and NLP for Use in River Flood Routing and Comparison with the Muskingum Method. 11th International Conference on Hydroscience & Engineering (ICHE) Hamburg, Germany. Zare, M. & Koch, M. 2016. Using ANN and ANFIS Models for simulating and predicting Groundwater Level Fluctuations in the Miandarband Plain, Iran. 4th IAHR Europe Congress. Liege, Belgium. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

A numerical groundwater flow model of Bursa Basköy aquifer

S. Korkmaz & G. E. Türkkan

Uludag University, Department of Civil Engineering, Bursa, Turkey

ABSTRACT: In this study, the aim is to determine the groundwater storage and distribution in the Basköy aquifer inside the city of Bursa, Turkey. Preliminary studies were conducted in the GIS environment, and afterwards, groundwater flow simulations were performed in MODFLOW which uses finite differences method to solve groundwater flow equations. From point elevation data, a digital elevation model (DEM) was constructed. The subbasins and drainage network were delineated through hydrological analysis of the DEM. The data were transferred to MODFLOW interface and a grid with a cell size of 15 m × 15 m was generated. Hourly groundwater level measurements at 7 observation wells were obtained from State Hydraulic Works. The observation data were used for calibration of unknown parameters such as hydraulic conductivity and storage coefficient. Calibration process was expedited by using the Parameter Estimation (PEST) model. Transient simulations were performed in daily timesteps and according to the results, model output was dependent on the initial condition, hydraulic conductivity and storage coefficient. In most of the wells the general dynamics of groundwater were successfully predicted.

1 INTRODUCTION

Basköy aquifer is located in the city of Bursa inTurkey. In order to investigate the hydrogeology of the region and the general behaviour of groundwater system field studies were performed by State Hydraulic Works of Turkey (SHW). According to these field studies, the geometry of aquifer, boundary conditions, sinks and sources such as wells and springs were all determined in order to identify the hydrogeologic properties of the formations (SHW 2013). Spring discharges were mea sured monthly. Groundwater heads were monitored using observation wells drilled at various locations on the basin. In addition to these, in order to identify the groundwater quality in the aquifer water samples were taken from wells, springs and fountains for chemical analysis.

In this study, a numerical model of the aquifer was developed and groundwater head distribution was solved by MODFLOW (McDonald and Harbaugh, 1988). MODFLOW solves 3D groundwater flow equa tions using method of finite differences. MODFLOW 2005 (Harbaugh, 2005) and MODFLOW-NWT (Nis wonger vd., 2011) are the two recent versions used in this study.

The data regarding the geometry of the basin was initially prepared in GIS (Geographic Information Systems) environment. The analysis performed are; acquisition of topographic maps; development of the digital elevation model (DEM); delineation of basin boundaries and drainage network using hydrological

analysis. Topographic elevations were transferred to

MODFLOW to generate a grid. 2 STUDY AREA Study area covers the region between Basköy, Ortaköy and Kadriye villages in the south of the city of Bursa, Turkey. The area has a size of about 5 km × 5 km. The field is distinguished from its surrounding as a raised plateau. Due to the vast existence of carbonated rocks and their dissolving property, a wide undulating karst topography is dominant in the area. Aquifer has a karstic structures with joints, fissures and fractures affecting the groundwater flow (SHW 2013). A total of 7 observation wells were drilled by SHW in order to determine the groundwater head distribution and the hydraulic properties of the aquifer. Automatic level recorders placed in these wells have been active since 27/12/2013. The wells are located upstream of Suyungözü spring which is the main water resource of the village of Basköy. There is a meteorological station in Kadriye village situated about 2 km away in the northeast of the area where daily rainfall and evapotranspiration measurements are made. In the current study, the data between 1/12/2013 and 31/12/2014 were used. In Figure 1, the map of the region prepared in GIS environment using WGS_1984 UTM_Zone_35N coordinate system is presented. 3 DEVELOPMENT OF THE NUMERICAL MODEL Initial studies for the development of the numerical model were conducted in GIS environment. An

Figure 1. Karstic field and observation wells in Basköy

region, Bursa, Turkey.

orthophoto map with a scale of 1:5000 and some dig

itized maps were obtained from Bursa Municipality.

These digitized maps included topographic contour

lines, buildings, roads and power lines. As only ele

vation data were to be used for the numerical model,

the remaining data were removed. First, point eleva

tions were picked over and saved as a separate file.

Then contour lines were transferred to another shape file and converted into 100 m-spaced point elevations. The two point elevation files were merged. In this way, all the available elevation data were used. As a result, thousands of point elevations were acquired all over the study area. These point elevations were then con verted to 40 m × 40 m resolution DEM in raster format using the ArcGIS programme. In order to delineate the boundaries of the basin, ArcHydro Tools programme was used and the following tasks were performed; fill sinks, find flow directions, compute drainage areas, delineate drainage network by giving a threshold to drainage areas, and finally, define sub-basin bound aries. As the field investigations of SHW covered the drainage area of the Suyungözü spring located southeast of the observation well KAR1, all of the sub basins falling in that area were included in the present study. A 3D view of the DEM of study area is depicted in Figure 2.

As can be seen from Figure 2, the study area appears as a raised plateau at elevations of 750–800 m.The data prepared in GIS environment were then transferred to GMS MODFLOW software. The governing equa tion of 3D groundwater flow solved by MODFLOW is defined as:

where K_{xx} , K_{yy} , K_{zz} are respectively the hydraulic con ductivity in x, y and z directions; W is the source term;

and S S is the storage coefficient. Figure 2. 3D view of the study area; cell size is 40 m × 40 m. As the basin area is smaller than the initial DEM, a higher resolution grid was generated in GMS MODFLOW interface. A grid frame with having a size of 3865 m×4546 m was drawn. Inside the frame, a grid with a cell size of 15 m×15 m was generated. The grid consisted of 258 cells in x-direction and 303 in ydirection. The cells falling out of the basin boundaries were made inactive. If a boundary condition such as constant head or general head is not specified to a cell falling on the boundary then the programme assumes an impervious boundary condition. Apart from these, two riverbeds were identified in the south of the basin and these were modelled using the river (RIV) package of MODFLOW. When using the RIV package, depth of water in the river, riverbed elevation and conductance values are specified. The direction of flux is decided by comparing the piezometric head to the river stage while the rate of flux is a function of conductance and the difference between the stage and piezometric head. In Figure 3, the starting point of the left river is where the Suyungözü spring is situated. Riverbed elevations were specified about 2 m below the surface elevation. Mean monthly spring discharges were specified as observed flow data. The wells used for groundwater head measurements were specified as observation wells. (Fig. 3). There were no physical boundaries nearby such as a river or lake. As subbasin boundaries represent local ridges and separate surface basins, in this study they were assumed to represent groundwater divides through which no flow occurs, and therefore, the entire aquifer boundary was modelled using impervious boundary condition. The types of simulations performed in this study are twofold, namely, steady-state and transient. The aim of the steady-state simulations is to obtain a general groundwater head distribution in the aquifer as well as the values of hydraulic conductivity. For this reason the annual mean of observed groundwater heads were used as observation data. As input, the annual mean of rainfall data were given to the model using the recharge (RCH) package. Simulations were performed using MODFLOW-2005. The parameter estimation (PEST)

Figure 3. Groundwater head distribution and error bars in

observation wells after steady-state simulation.

model (Doherty, 2013) was used for automatic calibra tion of parameters, mainly, hydraulic conductivity and storage coefficient. Initially, a total of 10 subbasins were delineated and they were used for calibration such that each subbasin was represented as a hydraulic conductivity zone. According to the results, the values were computed as 0.5 m/d in northern regions and up to 4 m/d in southern regions. The groundwater head distribution in the end of steady-state simulation is presented in Figure 3. It is observed that at highest elevations in the north, the groundwater head is above 800 m and it decreases with a steep slope down to 700 m in southern region.

In Figure 3, the scale of error bars is 1 m. When the residual is lower than 1 m, the colour is green. When it is between 1 and 2 m, the colour is yellow. If it is larger than 2 m, the colour is red. The model prediction of the groundwater heads at wells Kar4, Kar5 and Kar6 are close to the mean of observed values. In other wells, the model prediction falls below the mean observed values. Further calibration of hydraulic conductivity was needed in these regions and it was performed in transient simulations.

In transient simulations all the daily rainfall data between 1/12/2013 and 31/12/2014 were used. In addition, the evapotranspiration (EVT) package was enabled and the daily evapotranspiration values in the same period were input. In EVT package, an evapo transpiration surface elevation and an extinction depth were specified. The groundwater head distribution computed from steady-state simulation was specified as initial head distribution. MODFLOW-NWT solver was used in this case. The advantage of the NWT solver is that a stable solution can be obtained in non linear problems. Simulations were performed in daily Figure 4. Groundwater head time series for the observation well Kar1. timesteps within the period of available meteorological data. In transient calibration, the number of parameter zones were increased from 10 to 15 in order to obtain a more heterogeneous distribution and to better benefit from observation wells. It was observed that initial head distribution had an important impact on the overall error. Around some of the wells the initial head distribution were modified according to the first day of head observations. PEST model also computed parameter sensitivities during simulations. In some zones hydraulic conductivity had higher sensitivity and in others the storage coefficient. River conductance was also calibrated, however, it had a small sensitivity. Depending on the number of calibration parameters, a simulation took from 20 minutes to several hours on a 64-bit computer with 2 CPU’s and 20 cores at 2.3 GHz and 128 GB of RAM. Groundwater head time series for Kar1 is shown in Figure 4. According to the results, model prediction is dependent on initial head distribution, hydraulic conductivity and storage coefficient. 4 CONCLUSION The groundwater head distribution and characteristics of the Basköy aquifer in Bursa, Turkey were investigated using the numerical model MODFLOW. MODFLOW solves groundwater flow equations on a 3D grid using the method of finite differences. Preliminary studies for the determination of topographic surface and basin boundaries were conducted in GIS environment. A DEM was produced from point elevations, and afterwards, basin boundaries and drainage network were delineated using ArcHydro Tools programme. The data were then transferred to MODFLOW and a grid with a cell size of 15 m x 15 m was generated. Both steady-state and transient sim ulations were performed. The former was used to obtain a general groundwater head distribution and rough estimates of hydraulic conductivities by using mean annual rainfall. The groundwater table had a steep slope. Transient simulations were performed in between 1/12/2013 and 31/12/2014 using the PEST model which enhances the calibration process. EVT package was also enabled and daily evapotranspira tion values were used. In most of the wells the general dynamics of groundwater were predicted such as Kar 1 (Fig.4), however, sharp peaks in hydrographs are yet to be predicted. Those peaks in observations are thought to be due to the karstic nature of the aquifer. The results improved as the number of parameter zones increased.The most effective parameters in calibration were the initial head distribution, hydraulic conduc tivity and storage coefficient. PEST model facilitates calibration, however, it is not practical to calibrate all parameters in all zones at one single simulation due to computing limitations. Therefore, user judgement

Groundwater management and potential climate change impacts on Oum Er Rbia basin, Morocco

Mounia El Azhari & Dalila Loudyi

Water and Environmental Engineering, University Hassan II of Casablanca, Mohammedia, Morocco

ABSTRACT: The aquifers of Oum Er Rbia basin show a significant imbalance between the groundwater input and output because of their overexploitation for agriculture and drinking water uses. Indeed, in 2012, groundwater extractions reached 608 million m^3 while groundwater potential was only 347 million m^3, yielding a deficit of nearly 300 million m^3. Moreover, climate change will have a certain impact on the future of these resources. Recent studies predicted an increase in the mean annual temperature ranging between 0.1 and 1.4 $^\circ C$ by the period 2010–2030, and a decrease in the average annual rainfall of about 200 mm. The demand for irrigation water and drinking water will yet increase given the population growth coupled with progressive economic development. Climate change will adversely have an impact on aquifers recharge and the level of their water table, and consequently, the gap between supply and demand will increasingly rise. Groundwater models that take into account regional climate scenarios, are necessary to predict the potential impacts of climate change on groundwater resources sustainability. In this work, a reflection on a more efficient and sustainable management of groundwater resources in the Oum Er Rabia basin will be presented. The primary goal is to fill the gap between water supply and demand through climate change adaptation actions, in particular, the use of non-conventional water resources such as desalination of sea water, reuse of treated wastewater, rainwater harvesting, optimization of irrigation infrastructures and the adoption of projects of water transfer from other national basins with excess water.

1 INTRODUCTION

In many countries, groundwater is the main source for the supply of agricultural, domestic and industrial waters.Yet this vital resource, so important to survival, is facing several demographic, economic and environ mental pressures that threaten its availability and its durability.

According to the fourth assessment report of the Intergovernmental Panel on Climate Change, the cli mate in NorthAfrica should become even more hot and dry in the next century, exposing millions of people to water stress (IPCC, 2007).

2 CLIMATE CHANGE AND GROUNDWATER

Despite their importance and unlike surface water, groundwater has received little attention in terms of the assessment of climate change impacts (Parry et al., 2007). Nevertheless, there are studies that dis cussed the impacts of climate change on groundwater resources: Allen et al.(2004) studied the Grand Forks aquifer located in south central British Columbia in Canada to model the sensitivity of the aquifer to recharge and river levels changes connected to projected climate change scenarios; Sherif & Singh (1999) investigated possible impacts of climate change on saltwater intrusion in coastal aquifers; Holman (2006) described an integrated approach to qualify socio-economic impacts of climate change on the underground recharge in East Anglia in the UK. Morocco has a mobilized groundwater potential of nearly 4 billion m 3 spread over 130 aquifers (Fig.1) of which 98 are superficial and 32 are deep (DMCWater, 2014). However, the actual groundwater withdrawal volumes in 2014 are estimated to 5 billion m 3 per year (DMCWater, 2014). Groundwater exploitation is reaching saturation levels and the critical threshold is already reached due to overexploitation, sometimes dictated by the shortage or unavailability of surface water, while the demand is still growing. Moreover, studies addressing the future evolution of climate in the North African region show that the different used models predict the same trends in Morocco: an increase in temperature and a decrease in precipitation (Hulme et al., 2000; Driouech, 2009a; Mokssit, 2009; Born et al., 2008). The climate is very variable according to regions and seasons with a predominance of semi-arid Mediterranean climate.

Figure 1. Distribution of Groundwater Resources in

Morocco River Basins.

Figure 2. Situation of the Oum Er Rbia basin in Morocco.

3 OUM ER RBIA BASIN

The River Basin of Oum Er Rbia (RBOER), in addition to the Atlantic coastal basins of El Jadida-Safi, are spread over a total area of 48 070 km 2 , covering almost 7% of Morocco’s territory (Fig.|2). The main river, Oued of Oum Er Rbia, originates in the Middle Atlas mountains at 1800 m above sea level, and throws into theAtlantic Ocean near the city ofAzemmour crossing the chain of the Middle Atlas mountains, the Tadla plain and the coastal Meseta.

The RBOER has many regions with different cli mates: temperate coastal climate that characterizes the coastal zone, arid climate characterizing the Rhamna plain, semi-arid climate in the plain of Tadla and wet in mountain areas.

Precipitation in the basin decreases from West to East (1100-250 mm/year). However, in the upstream part of the basin, at high altitudes of the Atlas moun tains, the annual rainfall can reach 1000 mm, with a significant amount of snowfall at altitudes above 1500 m. The average precipitation in the basin is about 500 mm/year and varies between 1100 mm in the Mid dle Atlas and 300 mm downstream of the basin. The temperature varies between 10 and 50 $\circ$ C. The min ima and maxima are 3.5 $\circ$ C in January and 38 $\circ$ C in August The potential evaporation can reach 1600–1800 mm/year. On average, it is about 1600 mm/year in the coastal area and 2000 mm/year in the basin with a monthly maximum of 300 mm in July and August. Table 1. Water balance of different groundwater aquifers in the Oum Er Rbia basin (MPIWRM, 2012). The Oum Er Rbia basin is characterized by a large number of aquifers (11 aquifers) with a renewable groundwater potential of 350 million m 3 . Most of these aquifers have a large surface and are easy to recharge, which make them vulnerable to overexploitation. Their water balance as presented in Table 1 shows that almost all of the aquifers are unbalanced due to their overexploitation in a more or less important way depending on the area (MPIWRM, 2012). Indeed, the exploitable volume is 350 million m 3 , while withdrawals for irrigation and drinking water are 610 million m 3 , thus a deficit amounting to nearly 300 million m 3 (MPIWRM, 2012). 4 CLIMATE CHANGE IN THE OUM ER RBIA BASIN An analysis of changes in precipitation for the period 1980-2008 showed an average decrease of the annual rainfall in the area of 70 mm that is 20% relatively to the 1940–1980 period. During 2011–2013, the River Basin Agency of Oum Er Rbia conducted a series of studies with the support of the World Bank to describe climate evolution in the region by 2030/2060, and to characterize the possible impacts of climate change on water resources and on the balance between supply and demand during the period 2011–2040. Dynamical downscaling method was used to determine the future projections of climate, through three general circulation models (GCMs). Two models show a reduction in rainfall from 0.01 to 0.3 mm per day per decade in the period 1971 to 2065 and one model shows an increase in rainfall between 0.01 and 0.1 mm per day per decade over the same period. The three models agree on an increase in temperature, which varies between 0.05 ∘ C and 0.7 ∘ C per decade for the period 1971 to 2065. To study the past evolution of climate in the study area, we chose seven climatological stations to study the registered temperature changes and ten stations for rainfall changes.After tracing the history of the annual average temperature and precipitation for the different stations (Figs.|3 & 4), it is clear that climate trends in

Figure 3. Annual rainfall variation (1983–2014).

Figure 4. Monthly annual temperature variation

(1985–2014).

the region converge towards an increase in tempera

tures and decrease in precipitation with the alternation

of wet and dry years.

5 OBSERVED CHANGES IN THE BASIN

AQUIFERS

To assess the impact of these climate variations on

groundwater resources, we studied the evolution of

groundwater levels of some aquifers of the Oum Er

Rbia basin. The graphs show a significant drop in the

groundwater level (Fig.|5).

To better understand the relationship between the

variability of climate and water resources and to

clearly define the impact of climate change on ground

water, several factors must be determined such as:

the amplitude of the variation of temperature and precipitation compared to that of water levels, the aquifers response time to these variations and the estimation of recharge and evapotranspiration. Fur thermore, we should look out for other factors such as over withdrawal and growing demands that influ ence groundwater resources, especially if we assume that temperature and precipitation remain unchanged. Figure 5. Evolution of the groundwater level at few aquifers of the River Basin of Oum Er Rbia. 6 CLIMATE CHANGE ADAPTATION MEASURES IN THE OERHB At the Oum Er Rbia basin, the Master Plan of Integrated Water Resources Management was elaborated by the River Basin Agency of Oum Er Rbia in 2012 for a period of 20 years. However, this tool, used to manage and plan water resources until 2030, does not take into consideration climate change impacts to estimate future supplies and demands. For an integrated management of groundwater resources and to overcome the gap between supply and existing demand, proposing adaptation strategies

Figure 6. OCP desalination plants site map.

to climatic hazards scenarios proves to be an absolute necessity.Adaptation measures that have been adopted so far by the delegated ministry in charge of water, by the river basin agency, by the provinces or by the Cherifien Phosphate Office (OCP) are:

6.1 Desalination of sea water

At the RBOER, the Cherifien Phosphate Office (OCP) launched a program for the construction of two sea water desalination plants with a total capacity of 100 million m 3 /year (Fig.|6). These volumes will cover the future water needs of the Jorf Lasfar and Safi indus trial sites corresponding to 25% of the allocation in drinking and industrial water for 2030.

6.2 Collection and use of rainwater

The recovery and reuse of rainwater represent an alter native technique for water resources mobilization, especially for regions that are hard to reach. At the RBOER, a blueprint for the collection and the direct use of rainwater was established in 2011. It aims to have an idea about rainwater harvesting in the area, and to identify sites that could be developed and adapted to the regional context to collect these waters. The poten tial of rainwater harvesting in the basin is estimated at 1994 million m^3/year.

6.3 Water saving

Morocco has adopted a national irrigation water saving program (PNEEI). This program aims at the technical upgrading of irrigation systems and their adaptation to localized irrigation and the equipment of farms with irrigation drip system. In addition, the project of the slurry pipeline that transports the phosphate from Khouribga to the site of Jorf Lasfar hydrologically, allows water savings of up to 3 million m^3/year by eliminating drying operations. The recov ered water is reused in the phosphate wash process.

6.4 Reuse of treated wastewater

The use of non-conventional water resources, includ

ing the reuse of treated wastewater, is considered an alternative of water supply. Indeed, the purified wastewater can be used for agriculture, in industry, for watering green spaces or for recharging aquifers. At the RBOER, the reusable potential of treated wastewater in 2030 was estimated at 35 million m 3 / year. Currently, various studies for the reuse of treated wastewater are either complete or in progress, whereas some projects are already operational at Benguerir and Khouribga with a reusable volume of 5 million m 3 /year for industrial use, and at Boujaad with 0.77 million m 3 /year for agricultural use. 6.5 Water transfer from surplus basins The RBOER has nearly 600 km of water pipelines providing: – Intra basin transfers of 330 million m 3 /year from Bine El Ouidane dam for the irrigation of the downstream Tessaout. This transfer represents around 30% of available water resources and the remaining 70% are for the perimeter of Beni Moussa. – Extra basin transfers from Oum Er Rbia to several neighboring areas: 170 million m 3 /year for the supply of drinking and industrial water to Casablanca, Settat and Berrechid and 290 million m 3 /year for the irrigation of the Haouz plains (MPIWRM, 2012). With the impacts of climate change, the area will show a water deficit that requires external contributions from surplus basins to compensate the current transfers out of the basin. In this context, a large transfer project is under study. It will allow, through a network of 500 km, to transport a maximum flow of 45 m 3 /s of water from Sebou, Laou and Loukkos rivers to water deficit basins of Bouregreg, Oum Er Rbia and Tensift. 7 CONCLUSION The Oum Er Rbia basin groundwater is experiencing overexploitation. Water tables of different aquifers of the basin are dropping over the years. Regional climate scenarios predict an increase in temperature and decrease in precipitation by 2030, which will have inevitably a negative impact on groundwater resources. However, due to population growth and socio-economic development, excessive pumping and overextraction of water for domestic, industrial or irrigation use are the main causes of groundwater overexploitation. Future water strategies and plans have to integrate climate change scenarios to predict future supplies and demand. Other adaptation measures must be taken into consideration, in particular artificial groundwater recharge, exploration and mobilization of groundwater in mountainous areas, prohibition of irrigated perimeters extension and interdiction of new extraction demands.

Allen, D.M., Mackie, D.C., Wei, M. 2004. Groundwater and climate change: a sensitivity analysis for the Grand Forks aquifer, southern British Columbia, Canada, Hydrogeol ogy Journal, Vol. 12, pp. 270-290.

Born, K., P. Christoph, M. Fink, A.H. Knippertz, P. Paeth, and H. Speth, 2008. Moroccan Climate in the Present and Future: Combined View from Observational Data and Regional Climate Scenarios. In: Climatic Changes and Water Resources in the Middle East and North Africa. [Zereini, F. and H. Hötzl, H. (eds.)]. Springer, Berlin, Heidelberg, pp. 29-45.

Driouech, 2009a. Le changement climatique: Problématique mondiale/Réponse mondiale. Actes de la rencontre Inter nationale sur le Changement Climatique : Enjeux et perspectives d'adaptation pour le Maroc, organisée par l'Institut Royale des Etudes Stratégiques (IRES), Rabat le 16 Octobre 2009.

Holman, I.P. 2006. Climate change impacts on groundwa ter recharge – uncertainty, shortcomings, and the way forward, Hydrogeology Journal, Vol. 14, pp. 637-647.

Hulme, M., Wigley, T.M.L., Barrow, E.M., Raper, S.C.B., Centella, A., Smith, S. and Chipanshi, A.C. 2000. Using a Climate Scenario Generator for Vulnerability and Adapta tion Assessments: MAGICC and SCENGEN Version 2.4

Workbook, Climatic Research Unit, UEA, Norwich. IPCC, Climate Change 2007. Impacts, adaptation and vulnerability. Contribution of Working Group II to the Third Assessment

Report of the IPCC (ed. by McCarthy J.J., Canziani O.F., Leary N.A., Dokken D.J.,White K.S.), Cambridge University Press, UK, 1032 pages. Mokssit, A., 2009. Le Changement Climatique au Maroc: Observations et Projections. Rencontre Internationale sur le Changement Climatique: Enjeux et perspectives d'adaptation pour le Maroc. Institut Royal des Etudes Stratégiques, Rabat, 16 Octobre 2009. Parry, M.L., Canziani, O.F., Palutikof, J.P., van der Linden, P.J. and Hanson, C.E. (editors), 2007. Contribution of Working Group II to the Fourth Assessment Report of the Intergovernmental Panel on Climate Change. Cambridge University Press, Cambridge, UK. River Basin Agency of Oum Er Rbia, 2012. Impacts of Global Climate Change (GCC) on the Water Resources of Morocco, Phase 2: Hydrological Modeling for Oum Er Rbia River Basin and Phase 3: Integrated Water Resources Simulation for Oum Er Rbia River Basin. River Basin Agency of Oum Er Rbia, 2012. Master Plan of Integrated Water Resources Management (MPIWRM). Sherif, M.M., Singh, V.P. 1999. Effect of climate change on sea water intrusion in coastal aquifers, Hydrological Processes, Vol. 13, pp.1277–1287. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Flow modeling in vegetated rivers

M. Mabrouka

National Institute of Agronomy of Tunisia, University of Carthage, Tunis, Tunisia

ABSTRACT: The development of vegetation in river may increase flood risks. It plays an important role in influencing the hydrodynamic behavior, ecological equilibrium and environmental characteristics of water bodies. Therefore, it's important to have suitable prediction of increased resistance caused by vegetation. In the recent years, experimental and numerical models have both been developed to model the effects of submerged vegetation on open-channel flow. This paper describes a new analytic model based on physical equations used to predict hydraulic conditions in vegetated rivers. This model was programmed in two-dimensional software to

predict flow characteristic in vegetated flow. Simulations were applied on river reach of Medjerda in Tunisia. The comparison between simulated and observed results shows a good performance of this model in the prediction of flood in vegetated rivers.

1 INTRODUCTION

Vegetation development extends into the riverbed or in the secondary channel, could limit the initial flow area, it may affect the hydraulic conditions and hydrodynamic behavior of a stream. It could have a significant influence on increasing of bed roughness, on velocity distribution and on the water depth (Nepf & Vivoni, 2000, Liu & Shen, 2008, Mazaheri & Samani, 2009).

The vegetation exerted a drag force on flow, it reduced the shear stress and enhanced the local sediment deposition. The presence of vegetation within rivers changed the sediment distribution and disrupted by that the natural functioning of the stream (Klopestra et al., 1997, Lopez & Garcia, 2001, Defina & Bixio, 2005, Huthoff et al., 2007, wu & He, 2009, Morri et al., 2014).

Many rivers have been affected by vegetation problem. In Tunisia, the Medjerda, which was the only permanent river, was a subject for the development of intensive or extensive canopies (Zahar et al., 2008, Jaziri, 2009). In France, the Isere river showed a fixa tion and stabilization of vegetated islands areas which amplifies flood risks.

In Europe, North America and other parts of the world, rivers have been subject to serious disruptions due to the human activities. Many dikes were formed to prevent flooding, large dams were built and intense sediment extractions took place. These activities gen erated several problems such as the incision of the main channel and the lowering of the water level which led to the installation of the many vegetated islands and amplifies flood risks (Allain Jegou, 2002, Rodrigues et al., 2006, Jordain, 2013).

Thus, an understanding of vegetated flows is nec essary to control the floods and the ecosystem of the stream. Recently, much research has been devoted to understand flow characteristics in vegetated flow using flume experiments with natural or artificial vegetation and many numerical methods have been developed to describe flow – vegetation interactions. Some authors as jarvela et al., (2005) & Carollo et al., (2005) used a rectangular flume in laboratory to identify velocity distribution above flexible vegetation as the wheat. Others developed a numerical method to predict hydraulic parameters in a vegetated flow based on vegetation characteristics (vegetation height h p , density of vegetation m, diameter of plant stems D, drag coefficient C D), instead of using a constant roughness coefficient through analytical models (Tsujimoto et al, 1993, Shimizu & Tsujimoto, 1994, Klopstra et al., 1997, Lopez & Garcia, 2001, Stone & Shen, 2002, Jarvela, 2005, Baptist et al., 2007, Huthoff et al., 2007, Augustijn et al., 2008,Yang & Choi, 2010, Morri et al., 2014). In fact, Most of the developed relationships adopted a two-layer approach. This method based on dividing the flow domain into two layers. The first layer called “the vegetation layer” which is through vegetation, the other layer above it called “upper layer”. The different between these descriptions arise from the different assumptions used to determine the shear stress and the turbulent length scale. Some authors used the Boussinesq's eddy viscosity approach and the mixing-length theory to determine the shear stress and the velocity in each layer. The average velocity (U) over the total depth is given by combination between the mean velocity flow inside (U_1) and above the vegetation (U_2). In this paper, a new roughness model based on vegetation characteristics (Huthoff model) was included in a two-dimensional software Telemac 2D, to predict flow characteristic in presence of vegetation.

The validation of the programmed model was deter

mined through a comparison between measured data

in a 20 Km Medjerda reach and simulated one. Then,

the result of a vegetation maintenance scenario and its

effect on the water line was shown.

2 THEORETICAL BACKGROUND

Huthoff et al., (2007) derived an analytical model

for flow in presence of submerged vegetation. In this

model flow domain was divided into two layers.

In the vegetation layer the expression of the average

velocity (U_1) based on the momentum equation.

The momentum equation was given by the follow

ing expression:

where τ_{hp} = the shear stress at the interface and it's

determined using the following equation:

With ρ = the density of water (kg/m^3), g = the accel

eration gravity (m/s^2), h = the water depth (m), i = the

energy gradient, U_1 = the mean velocity in the vege

tation layer (m/s), C_D = the drag coefficient, m = the density of vegetation (m^{-2}), D = the stem diameter

(m), h_p = vegetation height (m).

The mean velocity is given by the equation (3):

With U_{r0} = the depth-averaged flow velocity in the

resistance layer for emergent resistance elements:

b = the drag length determined by the following equa

tion:

In the surface layer, Huthoff et al., (2007) derived

the expression of the average velocity based on the

condition of a constant energy dissipation rate from

large to smaller flow scale. Then the mean velocity

in the surface layer was determined by the following

expression:

With, s = the separation between individual resistance

elements: The average velocity over the total depth (U) is given by combination between the mean velocity flow inside (U_1) and above the vegetation (U_2): The expression for the average velocity of the entire flow depth becomes: This expression was programmed in a twodimensional code telemac2D using FORTRAN subroutines, to include the vegetation effect on the bed roughness and to determine an numerical method in the prediction of the hydrodynamic and sediment transport in presence of vegetation. Telemac 2D solves the Saint-Venant equations using the finite-element or finite-volume method and a computation mesh of triangular elements space based on the solution of the depth-averaged shallow-water equations. The Saint-venant equations solved by Telemac 2D were described with the following expression: Continuity equation: Momentum equations h = the water depth, Z = the free surface elevation, U = the mean velocity, F = the friction force, S = the bottom source term, Us = source term velocity and νe is the effective viscosity. 3 MODELING OF THE MEDJERDA REACH In Tunisia, the Medjerda is the only permanent river. This river traverses a length of 484 km including 150 km downstream of the Sidi Salem dam. It covers a total area of

23 700 km 2 (Fig. 1). The Medjerda is a typical Mediterranean river.). It was composed of three sub-watersheds: the high, middle and lower valley of the Medjerda. The upper valley corresponds to the highest part. This valley has its source in the mountains Medjerda to Algeria where the altitude can exceed the 1000 m and extends to the city of Ghardimaou 200 m. The average valley extends from the Sidi Salem dam ends near the town of El Arroussia. This watershed has an area of flood propagation and flood risk. It was seen several major floods, causing considerable material damage, and represent a real risk to local populations. However, it’s the floods of March 1973 and 2003 that remain etched in our memories.

Figure 1. The Medjerda river localization.

The river Medjerda, has known various water projects between 1982 and 1987 as the construction of dams, and hydraulic concrete thresholds to reduce the flood risk. Many authors noted that these constructions participated in the disruption of the natural functioning of the Oued (Zahar et al., 2008). Indeed, the existence of these structures led to a decrease in flow and causes the accumulation of solid materials, then limiting of the cross sections and extending of vegetation in the main channel and the banks.

TELEMAC modeling was applied to a section of the Medjerda of 19 km in length and 75 m wide.

The chosen study area is located in the middle val ley and characterized by several meanders. Available bathymetric, topographic and hydraulic data needed to define the study area. It is proposed to study the flood Medjerda (February–March) 2012.

The aim of this modeling is the verification of the ability of the programmed model in predicting the hydrodynamic real case and evaluating the effect on the vegetation of the hydraulic parameters and propagation of flood.

In order to model flood risks in the studied reach, available databases were exploited for the model implementation, such as the digital elevation model, and topographic data maps. I also used databases mea sured during the floods of February 2012 such as flood hydrographs, water depth and the rating curve. For veg etation species, in the Medjerda, the Tamarix africana and Nerium Oleander were the more developed on the banks, However, the flow channel is invaded by aquatic plants and semi-aquatic, mainly Typha angustifolia.

This vegetation grows in the minor and major bed

Medjerda. It is characterized by rapid proliferation. Figure 2. Meshing studied reach. The average height of the vegetation is about 10 cm, with an average diameter of 6 mm, the longitudinal and transverse spacing is 5 cm and the density of this species in the studied section is about 400 m^{-2} . The studied field is meshed with triangles on a nonstructured grid of about 23762 nodes. The meshing software called blue kenue with a fine mesh near river bed on the edges of the study area (Fig. 2). After the meshing of the studied reach, I defined I the boundary conditions based on the available hydraulic data. Upstream of the system, I imposed the flood hydrograph 2012 observed at Slouguia station, the liquid boundary in this case is the free depth and the imposed velocity U and V. In the Downstream, of

Figure 3. flood dynamic comparison.

the studied reach I impose a rating curve, character ized by an imposed depth and velocity. The banks are characterized by a solid wall.

4 SIMULATION RESULTS

4.1 Flood dynamic simulation

The Figure 3 shows a comparison between the mea sured flood hydrograph of 2012 and the simulated one using the new approach of Huthoff. I realized an approximate hydrograph based on the rating curve for comparison due to the luck of data measured down stream of the study area, as the hydrometric station at Medjez el Bab is damaged. The overall shape of the flood is correct. Note that the peaks of the sim ulated flood fall on the same dates as those of the measured raw, this confirms that the error relative to the propagation time is almost zero.

However, a discrepancy can be seen between the two hydrograph of about 20 m^3/s. The discrepancy observed between measured and simulated results could be explained by the using of data for the cross sections of 2003, whereas the hydrodynamic flood dad was of the 2012 (linked to lack of data).

4.2 Water depth simulation

According to various studies of Fdhila, 2006; Sendi, 2010; Amara, 2012 and Gharbi, 2016, the roughness coefficient of Medjerda river is about 20 m 1/3 /s. I car

ried out a comparison between the water lines obtained

by the programmed model using the Huthoff Telemac

2D (Figure 4), and that by 1D software (Mike 11).

The two simulations show a small difference between

the water levels. This comparison allows testing the

capacity of th new model in predicting the water line

and emphasizes the validity of the model.The obtained

results by these two models show also that the over

flowing section is located in the city of Medjez al Bab.

Actually This town was affected during the flood of

2012. Figure 4. Water depth comparison. Figure 5. Water depth comparison with and without vegetation. 4.3 Water line comparison in the presence and without vegetation shows a comparison between the waterline with and without the presence of vegetation in the studied section. The presence of vegetation in the stretch of Slouguia-Medjez al Bab caused an increase in water level. The discrepancy between the water lines is about 1 m. Vegetation increases the roughness of the base and limits the water flow cross-section and accordingly the water level. This simulation shows the utility of the removal of vegetation installed at the bottom and banks to reduce flood risk. 5 CONCLUSION The verification of the performance of the new model is determined due to a comparison between the simulation results and the measured data of flood 2012. This verification shows the performance of the model in the prediction of the hydrodynamic flood and the surface line. Flows rates are in agreement with those observed. The simulated flood peaks are located in the same measured raw dates, this confirms that the error relative to the propagation time is almost zero. The peaks of the water depth also fall on the same dates. The model show also a good results in the prediction of the water level through a comparison with a simulated results carried out using 1D model. A scenario of maintenance of vegetation, demonstrated the influence of vegetation in the water line. The presence of vegetation in the stream increases the resistance to flow and reduce the shear stress, which lead

to reducing of
the velocity and the increasing of the water line, and affects the flood time propagation.

Allain Jégou, C. 2002. Relations végétation, écoulement transport solide dans le lit des rivières, étude de l'Isère dans le Grésivaudan. Thesis memory, Laboratory for the study of transfers in hydrology and environment, National Polytechnic Institute of Grenoble.

Augustijn, D.C.M., Huthoff, F. &VanVelzen, E.H. 2008 Com parison of vegetation roughness descriptions. Presented at the International conference on Fluvial Hydraulics, River flow, ÇeS¸ me. Izmir, Turkey, pp. 3–8.

Baptist, M.J., Babovic, V., Uthurburu, J.R., Keijzer, M., Uittenbogaard, R.E., Mynett, A., Verwey, A. 2007. On inducing equations for vegetation resistance, .J. Hydraul. Res.,vol. 45, no. 4, pp. 435–45.

Carollo, F.G., Ferro, V. & Termini, D., 2005. Flow resis tance law in channels with flexible submerged vegetation. Journal of Hydraulic research 131, 554–564.

Defina, A & Bixio, A.C. 2005. Mean flow and turbulence in vegetated open channel flow, Water Resour. Res., vol. 7, pp. 1–12.

Huai, W.X., Zeng, Y.H., Xu, Z.G., Yang, Z.H. 2009. Three Layer Model for vertical velocity distribution in open channel flow with submerged rigid vegetation, Adv. Water

Resour., vol. 32, no. 4, pp. 487–492.

Huthoff, F., Augustijn, D.C.M & Hulscher, S. J. M. (2007). Analytical solution of the depth-averaged flow velocity in case of submerged rigid cylindrical vegetation. Water Resour. Res., vol. 43, pp. 1–10, 2007.

Jaballah, M. 2009. Modélisation hydro-géomorphologique et cartographie du risque d'inondation. Master Thesis, hydraulic modeling laboratory and environment, National School of Tunis engineer.

Jarvela, J. 2005. Effect of submerged flexible vegetation on flow structure and resistance, J.Hydrol., vol. 307, pp: 233–241.

Jaziri, B. 2009 .Le bio et le barrage, végétation riveraine et aménagement hydraulique (eau libre, eau contrainte), structure, fonctionnalité, représentations dans trois cours d'eau de la Tunisie septentrionales, thesis, Ecole Normale Superior letter and Human Science, Tunisia.

Klopstra, D., Barneveld, H. J. van Noortwijk, J. M & Van Velzen, E. H. 1997. Analytical Model for Hydraulic Roughness of Submerged Vegetation . Presented at 27th International congress, IAHR, Managing Water: Coping with Scarcity and Abundance, San Fransisco, New York,

pp. 775–780. Liu, C & Shen, Y-M. 2008. Flow Structure and Sediment Transport with Impacts of Aquatic Vegetation,"J. Hydrodyn., vol. 20, no. 4, pp. 461–468, 2008. López, F and M.H. García, M.H. 2001. Mean Flow and Turbulence Structure of Open Channel Flow through Nonemergent Vegetation, J. Hydraul. Eng., vol. 127, no. 5, pp. 392–402. Maridet, L.1995. Rôle des formations végétales riveraines. Recommandations pour une gestion régionalisée. Final report. Convention No. 1/94. Cemagref, Bely/LHQ. Minister of the Environment. SDMAP PARIS, p. 69. Morri, M., Soualmia, A & Belleudy, Ph. (2014). Analytical Models Evaluation for Predicting Vertical Velocity Flows through Submerged Vegetation. International Journal of Engineering Research Volume No. 4, Issue No. 1, pp: 9–12. Nepf, H. & Vivoni, E. R. 2000. Flow structure in depthlimited, vegetated flow, journal of geophysical research, vol. 105, no. c12, pages 28, 547–28, 557. Rodrigues, S., Bréhéret, J.G., Macaire, J.J., Moatar, F., Dana Nistoran, D., Philippe Jugé, ph. 2006. Flow and sediment dynamics in the vegetated secondary channels of an anabranching river: The Loire River (France). Sedimentary Geology 186 (2006) 89–109. Samani, J.M.V. & Mazaheri, M. 2009.An analytical model for velocity distribution in transition zone for channel flows over inflexible submerged vegetation, J. Agr. Sci. Tech., vol. 11, pp. 573–584. Shimizu, Y. & Tsujimoto, T. 1994. Numerical Analysis of Turbulent Open-channel Flow over a Vegetation Layer Using a k-ε Turbulence Model. J. Hydrosci. Hydraulic. Eng., 11(2): 57–67. Stone, M.B. and Shen, H.T. (2002). Hydraulic Resistance of Flow in Channels with Cylindrical Roughness, J. Hydraul. Eng., vol. 128, no. 5, pp. 500–506, 2002. Tsujimoto, T., Okada,T & K, Kontani, K.(1993). Turbulent structure of open-channel flow over flexible vegetation," Progressive Report. Hydr. Lab., Kanazawa University, Japon, pp. 37–45. Wu, W & HE, Z. 2009. Effects of vegetation on flow conveyance and sediment transport capacity," IJSR, vol. 23, no. 3, pp. 247–259. Yang, W & Choi, S. 2010. A Two-layer approach for depthlimited open-channel flows with submerged vegetation, J. Hydraul. Res., vol. 48, no. 4, pp. 466–475. ZaharY., Ghorbel A., Albergel J. 2008. Impacts of large dams on downstream flow conditions of rivers:Aggradation and reduction of the Medjerda channel capacity downstream of the Sidi Salem dam (Tunisia). Journal of Hydrology 351, 318–330. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Comparison of methods to calculate the shear velocity in unsteady flows

G. Bombar

Ege University, İzmir, Turkey

ABSTRACT: The shear velocity is a significant parameter in all areas of hydraulics; therefore it is essential to

calculate it correctly. There exist various approaches to obtain the shear velocity in steady flows, but the literature

on the application of these procedures to unsteady flows is scarce. In this study, an artificial triangular-shaped

hydrograph was generated in a rectangular flume of 70 cm wide and 18 m long. The time series of the shear

velocity was obtained by several methods and compared, such as, u_{*SV} by applying the Saint-Venant equations,

u_{*L} by adopting the methodology given by Clauser Method, u_{*P} by employing the parabolic law, u_{*UN} by applying

the momentum equation presuming the slope of energy grade line is equal to bed slope and finally u_{*avg} by using

the average velocity equation. In order to examine the hysteretic behavior of depth variation with point velocity, a

hysteresis intensity parameter η was proposed. Looking at the variation of η with normalized elevation as z/h_{base},

one can observe that there is a strong relation between these two dimensionless parameters and an equation was

proposed. After an analysis of existing literature data, it is revealed that the coefficient and power of the equation

depends on the unsteadiness parameter α.

1 INTRODUCTION

Flow parameters such as velocity, turbulence and

boundary shear stress in steady flow conditions were

studied widely both experimentally and numerically.

However, in nature, unsteady flows are the most

common type of open channel flows and attracted

a great amount of interest for research in the field

of hydraulics (Bose and Dey 2012). The unsteady flow experiments in open channels were conducted on hydraulically rough surfaces by Tu (1991), Song and Graf (1996), Qu (2002) and Bares et al. (2008) and on smooth surfaces by Nezu et al. (1997). During the ris ing and falling phases of the hydrograph, the validity of logarithmic law distribution was investigated (Nezu and Sanjou 2006). The flow features characteristics like average velocity, turbulence intensity, Reynolds stress and shear velocity are other issues examined in unsteady flow conditions.

According to Nezu (2005) and Nezu and Nakagawa (1993), in steady flows the shear velocity can be cal culated by various methods, such as, by the use of Reynolds stress graph, velocity data at the viscous sub-layer, Clauser method, parabolic law, the slope method where Saint-Venant equations are used and average velocity equation. Under unsteady flow con ditions, some researchers prefer to estimate the shear stress using the depth-slope product rule correspond ing to normal (steady and uniform) flow especially when the purpose of their study is to focus on over all parameters rather than local ones (Hassan et al. 2006, Güney et al. 2013) or prefer to neglect the con tribution of velocity gradient to energy slope (Powell et al. 2001), but the comparison of these methods is limited (Bombar, 2016). Recently Mrokowska et al. (2015a) used measurements obtained from artificial dam-break flood waves in a small lowland watercourse to analyze the resistance parameters in unsteady flow. Meanwhile, Mrokowska et al. (2015b) estimated the friction velocity using two equations that have been derived from two dimensional Reynolds and onedimensional Saint-Venant equations using data with high resolution obtained at laboratory. This study was carried out in the laboratory with artificial triangular-shaped hydrographs. Among the methods that have been developed to estimate the shear velocity (u_*), u_{*SV} the Saint-Venant equations, u_{*L} the methodology given by Clauser Method, u_{*P} the parabolic law and u_{*UN} the momentum equation assuming the bed slope is equal to the slope of energy grade line and the equation for average velocity u_{*avg} are used which are briefly explained in this paper. Furthermore the hysteresis was investigated and a hysteresis intensity parameter η is proposed as the area between the limbs of instantaneous point velocity vs. flow depth graph, normalized by the difference of peak and base values of the unit discharge. It is then related to the unsteadiness parameter α, in terms of time averaged instantaneous velocity variation with flow depth. The intensity of the hydrograph can be represented by an unsteadiness parameter, α which is non-dimensional and can be defined as the change of flow depth or flow rate within a specified time interval (Nezu et al. 1997):

where $\Delta h = h_{peak} - h_{base}$, h_{base} and h_{peak}, are the base

flow depth and maximum flow depth defined here

as peak flow depth, $V_c = (V_{base} + V_{peak})/2$, V_{base} and

V_{peak}, are the base flow and peak flow average veloci

ties, T_{rQ} is the rising duration of the hydrograph based

on the flow rate.

2 EXPERIMENTAL SET-UP

The experiments were carried out in a 70 cm wide,

18 m long rectangular flume with a bed slope of

0.004 in the Hydraulics Laboratory of Ege Univer

sity, Department of Civil Engineering. A tail gate of

25 cm high was located at the downstream of the flume. The bed material used in the flume was composed of a non-uniform sediment mixture with geometric standard deviation $\sigma_g = 2.27$ and $d_{50} = 0.43$ mm. At the first 1.7 m of the flume coarse grains were placed in order to prevent the local scour at the entrance. No sediment transport was observed during the base and peak flow conditions.

The water was circulated continuously. The pump used in this study was capable of producing a maximum discharge of 100 l/s, and it was connected to a pump speed control unit (PSCU) that regulate the flow rate by a program via increasing and/or decreasing the pump speed at predetermined time increments. An electromagnetic flow meter (Optiflux by Krohne) was affixed on the pipe before the entrance of the channel in order to measure the flow rate with a precision of 0.01 l/s. The water depths were measured by means of ultrasonic level meters (IMP+) with a precision of 0.1 mm which were placed 6 m, 6.75 m, 8.3 m, 9.1 m, 10.5 m and 11.25 m from the upstream end of the flume. The IMP+s and the flow meter were connected to a data logger (by Brainchild) which can record the data instantaneously.

The point velocities were measured by an ultrasonic instrument, Flow Tracker (FT by Sontek) which was located at 9 m from the entrance of the channel. So as to obtain the velocity profile, the hydrograph was repeated at each run, while changing the elevation of the instrument along the vertical axis. The instanta neous time series of velocity components were mea sured at 10 elevations, which means the hydrograph was repeated 10 times.

All experiments were recorded by a camera and a chronometer was used in order to check and validate the synchronization of the instruments and the PSCU.

3 PRELIMINARY RESULTS

A triangular-shaped asymmetrical hydrograph was generated in the flume.Also the data of Bombar (2016) is used in which three triangular – shaped hydro graphs with same base and peak pump speeds but different rising durations were generated similar to the one presented in this study. The rising durations were

90 s, 180 s, 270 s and 540 s, for H1 (present study), Figure 1. Flow depth and discharge variation of H1 with time. Figure 2. Variation of the water surface profile with time. Exp1, Exp2 and Exp3 (Bombar, 2016), respectively. The falling duration of all was 60 s. The flow rate was increased gradually to its steady base flow value then the hydrograph was started ($t = 0$). The coordinate x is defined as the distance from the entrance of the flume, y is the transverse distance from the center line which is the line of symmetry of the center line of flume, and z is the vertical distance from the surface of the bed. The instantaneous point velocities u and w in x and z directions, respectively, were decomposed into time varying mean point velocities $\bar{u}$ and $\bar{w}$ and their time varying

fluctuating components of point velocities u' and w' as $u = \bar{u} + u'$ and $w = \bar{w} + w'$ by moving average algorithm (Bombar et al. 2010). The smoothed velocity time series was used only in calculating the partial derivative of cross sectional mean velocity with respect to time $\partial V/\partial t$ in Saint-Venant method and to obtain the time that the parameter attains its maximum value accurately which are given in Table 1. In the rest of the calculations, the raw velocity time series was used. The discharge and flow depth variation with time t of H1 is depicted in Fig. 1. The variation of the water surface profile is given in Fig. 2 for H1. The profiles are given at times 0.0, 0.2, 0.4, 0.6, 0.8 and 1.0 times T_r based on the flow depth of the rising period. The properties of the experiments are summarized in Table 1. The average velocity at any instant time V is calculated by integrating the instantaneous point velocities along the water column. Here Q is the flow rate, Q_{base} and Q_{peak} , are the base discharge and peak

Table 1. Properties of the experiments, present study and experiments of Bombar, (2016).

Parameter	H1	Exp1	Exp2	Exp3
Q_{base} (l/s)	2.1	2.1	2.1	2.1
Q_{peak} (l/s)	38.2	38.8	38.4	39.3
Q (l/s)	36.1	36.7	36.4	37.3
T_{rQ} (s)	9 6	187	273	544
h_{base} (cm)	12.2	12.2	12.2	12.2
h_{peak} (cm)	19.5	20.1	20.6	20.8
h (cm)	7.3	7.9	8.4	8.6
T_r (s)	108	194	282	553
T_f (s)	230	218	230	210
$T = T_r + T_f$ (s)	338	412	512	763
V_{base} (cm/s)	2.4	2.4	2.4	2.4
V_{peak} (cm/s)	25.5	25.7	25.9	26.9
V_c (cm/s)	14.0	14.1	14.2	14.7

α 0.005 0.003 0.002 0.001

discharge, $\Delta Q = Q_{peak} - Q_{base}$, T_r and T_f are the rising

and falling durations of the hydrograph based on the

flow depth and $\Delta T = T_r + T_f$.

4 CALCULATION OF SHEAR VELOCITY

In order to calculate the shear velocity methods for

calculating the shear velocity, the momentum equa

tion (u_{*UN}), the Saint-Venant method (u_{*SV}), Clauser

Method (u_{*L}) the Parabolic Law (u_{*P}), and (u_{*avg}) are

used.

4.1 The Slope Method, Saint-Venant Equations (u_{*UN} & u_{*SV}):

In steady uniform flows, from momentum balance u_*

is related to gravitational acceleration g, channel slope

S_0 and R_h as;

The shear velocity estimated by Eq. (2) is abbrevi

ated as u_{*UN} .

In unsteady flows the u_* becomes $u_* = (ghS_e)^{1/2}$

where S_e is the slope of energy grade line. Saint

Venant equations consist of continuity and momentum

equations as given in Eq. (3.a) and in Eq. (3.b),

respectively.

where A is the cross sectional area and $S_0 = -\partial z/\partial x$.

After mathematical manipulations, one can get; Figure 3. Time variation of the terms in Eq. (5). Table 2. Contribution of the terms in Eq. (5). Contribution (%) H1 Exp1 Exp2 Exp3 S_0 48.5 50.1 50.4 50.6 $-\partial h/\partial x$ 46.9 46.7

47.0 47.4 $(V^2/gh)\partial h/\partial x$ 0.8 0.7 0.7 0.7 $(V/gh)\partial h/\partial t$ 0.9 0.5 0.4 0.3 $-(1/g)\partial V/\partial t$ 2.9 2.0 1.6 1.0 or, The time variation of the first, second, third, fourth and fifth terms in parenthesis of Eq. (5) which are S_0, $-\partial h/\partial x$, $(V^2/gh)\partial h/\partial x$, $(V/gh)\partial h/\partial t$ and $-(1/g)\partial V/\partial t$, respectively are given in Fig. 3. It is seen that the magnitude of the third and fourth terms are much less than other terms. The contribution of each term on u_* is given in Table 2. Obviously the S_0 and spatial-variation of flow depth have the dominating roles as mentioned by Qu (2002). The shear velocity estimated by Eq. (5) is abbreviated as u_{*SV}: 4.2 Clauser Method (u_{*L}): Nikuradse (1933) has proposed a logarithmic law to describe the vertical distribution of u. The shear velocity may be obtained from the slope of the best fit line in the inner region where $\chi = z/h < 0.2$ (Graf and Altınakar, 1998). When $u_* k_s/\nu < 5$ the flow is assumed to be smooth and the Eq. (6) is valid. where k_s is the Nikuradse's equivalent sand roughness, κ is the von Karman constant and can be taken as 0.40, B_s is the integral constant for smooth boundaries. The rough regime occurs when $u_* k_s/\nu > 70$ and the Eq. (7) is valid as,

Figure 4. Variation of f and B_r with time.

where z_0 is the reference level, B_r is the integral constant for rough boundaries.

The k_s is taken as a constant related to a representative sediment diameter, such as d_{50}. The reference level for a completely rough bed was taken as $0.033k_s$ by Jan et al. (2006) and $-0.25k_s$ by Song and Graf (1996). Brereton et al. (1990), Tu and Graf (1992), Tardu et al. (1994) and Song and Graf (1996) investigated the velocity profile for unsteady flows and concluded that the logarithmic law is valid in the inner region of open channel flow. Song and Graf (1996) find that the log law is valid in the inner region and Coles wake law is valid in the outer region. Nezu and Nakagawa (1991) mentioned that for high Reynolds numbers this devi

ation from log law is more prominent. It appears that under the conditions of the present study, the loga

rithmic law could be applied to determine the shear velocity.

For unsteady flows, Nezu et al. (1997) determined that the von Karman constant is not considerably affected from the unsteadiness.

The integration constant for smooth boundaries B_s has an average value of 5 (±25%). In unsteady flows, Onitsuka and Nezu (1999) and Nezu and San

jou (2006) claimed that for hydrographs with small unsteadiness values ($\alpha \approx 0.001$), the B_s remains nearly constant throughout the hydrograph, but for the ones with higher unsteadiness, ($\alpha > 0.001$) B_s increases in the rising period and decreases in the falling period.

The integration constant for rough boundaries B_r has an average value of 8.5 (±15%). Song and Graf (1996) revealed that the B_r parameter is equal to 8.5 as an average value. Tu and Graf (1992) calculated B_r in the range of 3.8–14.5. Similarly Song (1994) used the velocity data in the inner region to calcu

late the B_r and concluded that in rising limb B_r has smaller values than the one in the falling limb. Varia tion of friction coefficient f and B r with time is given in

Fig. 4.

The linear best fit trend line of u versus ln(z)

was obtained for each velocity profile measured at

each sampling time as performed by Qu (2002), Nezu

and Sanjou (2006) and Tu (1991). The shear velocity

estimated by Clauser method is abbreviated as u_{*L} . Figure 5. Variation of shear velocities u_{*UN} , u_{*SV} , u_{*L} , u_{*P} and u_{*avg} for experiment H1 with time. 4.3 Parabolic Law (u_{*P}): The parabolic law is given by Eq. (8) and applicable for the velocity data in the outer region (Graf andAltınakar 1998). where u_m is the maximum velocity, λ is given by the equation below (Afzalimehr et al. 2007) Taking χ = 0.2 will make the λ= 7.8. The slope of the linear trend line between u and $(1 - z/h)^2$ can be used to calculate the u_* . The shear velocity estimated by Eq. (8) is abbreviated as u_{*P} . 4.4 Average velocity (u_{*avg}): The equation given below is used to calculate the shear velocity as u_{*avg} . The Eq. (10) is used to find the shear velocity by using the mean average velocity V . The equivalent sand roughness was taken as $k_s = 10 d_{50}$. The time variation of the obtained shear velocities named as u_{*UN} , u_{*SV} , u_{*L} , u_{*P} and u_{*avg} are depicted in Fig. 5. The shear velocity u_{*UN} was overestimated due to the steady and uniform flow assumption, indicating the application of steady flow approximation is not correct (Mrokowska et al. 2015a). The u_{*SV} was also slightly higher than the others, particularly during the first phases of the hydrograph. On the other hand u_{*P} and u_{*avg} coincides well. As a conclusion the parabolic law and the average velocity equation can be used to calculate shear velocity interchangeably (Bombar, 2016).

Figure 6. Hysteresis between flow depth and mean cross

sectional velocity.

Figure 7. Hysteresis between flow depth and B_r .

5 HYSTERESIS

In unsteady flows it is a well-known fact that there

is a hysteresis between flow depth h and mean cross sectional velocity V where the velocity reaches its maximum value before the flow depth as depicted in Fig. 6 (Graf and Altınakar 1998). Also the hysteresis observed in flow depth and B r is given in Fig. 7. Considering the data of Bombar (2016), it is revealed that there is a clock-wise hysteresis. For experiments with small unsteadiness values, the rising and falling limbs are close to each other whereas for other experiments the limbs become further apart.This distance between the limbs depends on the unsteadi ness of the hydrograph as mentioned by Graf and Altınakar (1998).

A hysteresis intensity parameter η which was pro posed by Bombar (2016) is used for present study in order to calculate how far the limbs are from each other. The area between the limbs is normalized by the difference of unit discharges those corresponding to peak and base values, q peak and q base , respectively. It is revealed that the hysteresis intensity parameter η increases with flow depth, in other words closer to the surface the more the hysteresis is felt.The variation

of hysteresis intensity parameter η with normalized Figure 8. The variation of hysteresis intensity parameter η with normalized elevation z/h base (Bombar, 2016). Figure 9. Variation of coefficient and power in Eq. (12) with α. elevation i.e. z/h base , is shown in Fig. 8. The trend-line reveals the relation between these two dimensionless parameters and the equation for H1 is as follows, It is also concluded that the constant of the power equation for H1 is a = 0.17 and it is higher than those obtained for Exp1, Exp2 Exp3, 0.14, 0.13 and 0.11, respectively. This reveals that the more the unsteady the hydrograph, the more the hysteresis η. The relationship between the coefficient a and power b of Eq. (12) and the unsteadiness parameter is searched. It is observed that there is a strong dependence of η on α, as given in Fig. 9. 6 CONCLUSION The shear velocity is a crucial parameter in characterizing the shear at the boundary and there are methods to evaluate the shear velocity in steady flows. These methods are listed as the use of the Saint-Venant equations u_{*SV}, use of the procedure given by Clauser Method u_{*L}, use of the parabolic law u_{*P}, use of the momentum equation assuming the slope of energy grade line is equal to bed slope u_{*UN} and use of the average velocity equation u_{*avg}. The methods which are used to calculate the shear velocity in steady flows

were tested under unsteady flow conditions for an asymmetrical triangular-shaped hydrograph H1 and the data of Bombar (2016).

The stream-wise components of point velocity time series were measured by an acoustic Doppler veloc ity meter. The instantaneous velocity time series were obtained at various vertical elevations and the mov ing average algorithm is adopted to retrieve the time varying mean velocity. The variation of flow depth at various locations along the flume was used to calculate the water surface and energy slope variation. Eq. (5) is obtained by which the u_{*SV} can be calculated. It is seen that S_0 and $(-\partial h/\partial x)$ have the dominating roles. The momentum equation assuming the flow is uniform was employed to calculate the shear velocity u_{*UN}. The shear velocity was also calculated as u_{*L} by using the

Clauser method. Since the logarithmic law is assumed to be valid, this result is limited to the conditions of the present study. The shear velocity calculated by using the parabolic law is denoted by u_{*L} . The well-known average velocity equation was used to obtain the shear velocity as u_{*avg} .

Among the methods, u_{*UN} and particularly during the first phases of the hydrograph u_{*SV} were higher than the others. On the other hand u_{*P} and u_{*avg} coin cides well. It is concluded that the shear velocities calculated by the parabolic law and the equation of average velocity can be utilized interchangeably.

The hysteresis is investigated and the depth vari ation of hysteretic behavior of point velocity was observed. It is revealed that the hysteresis intensity parameter η increases with flow depth, in other words the closeer to the surface the more the hysteresis is felt.

ACKNOWLEDGEMENTS

The author would like to express her gratitude to Assoc. Prof. Ş̧ebnem ELÇİ for supplying the ultra sonic velocity meter used during the execution of the experiments.

Afzalimehr, H., S. Dey and P. Rasoulianfar (2007). Influence of decelerating flow on incipient motion of gravel-bed stream. Sadhana, 32(5), 545–559.

Bares, V., J. Jirak and J. Pollert (2008). Spatial and temporal

variation of turbulence characterisitcs in combined sewer

flow. Flow Measurement and Instrumentation, 19(3–4),

145–154.

Bombar, G. (2016). The Hysteresis and Shear Velocity in

Unsteady Flows. Journal ofApplied Fluid Mechanics 9(2),

839–853.

Bombar, G., M. S¸. Güney, G. Tayfur and ª. Elçi, (2010). “Cal

culation of the time-varying mean velocity by different

methods and determination of the turbulence intensities”,

Scientific Research and Essays, 5(6), 572–581.

Bose, S. K. and S. Dey (2012). Turbulent unsteady flow

profiles over an adverse slope”, Acta Geophysica, 61(1),

84–97. Brereton, G.J., W. C. Reynolds and R. Jayaraman (1990). Response of a turbulent boundary layer to sinusoidal free-stream unsteadiness. Journal of Fluid Mechanics, 221, 131-159. Graf, W. H. and M. S. Altınakar (1998). Fluvial Hydraulics, John Wiley & Sons Inc. Güney M. S¸., G. Bombar, A. Ö. Aksoy (2013). Experimental study of the coarse surface development effect on the bimodal bed load transport under unsteady flow conditions. Journal of Hydraulic Engineering, 139(1), 12–21. Hassan, M.A., R. Egozi and G. Parker (2006). Experiments on the effect of hydrograph characteristics on vertical grain sorting in gravel bed rivers. Water Resources Research, (42), 1-15. Jan, C. D., J. S. Wang and T. H. Chen (2006). Discussion of Simulation of flow and mass dispersion in meandering channel. Journal of Hydraulic Engineering, 132(3), 339–342. Mrokowska, M. M. Rowinski, P. M. and Kalinowska M. B. (2015a). A methodological approach of estimating resistance to flow under unsteady flow conditions Hydrol. Earth Syst. Sci., 19, 4041-4053, DOI:10.5194/hess-194041-2015.

Mrokowska, M. M., Rowiński P. M. and Kalinowska M. B. (2015b) Evaluation of friction velocity in unsteady flow experiments, Journal of Hydraulic Research, 53:5, 659–669, DOI: 10.1080/00221686.2015.1072853 Nezu, I. (2005). Open-channel flow turbulence and its research prospect in the 21st century. Journal of Hydraulic Engineering, 131(4), 229–246. Nezu, I., and H. Nakagawa (1991). Turbulent structures over dunes and its role on suspended sediments in steady and unsteady open-channel flows. Proc. of Int. Symp. on Transport of Suspended Sediments and its Mathematical Modeling, IAHR, Firenze, 165–189. Nezu, I. and H. Nakagawa (1993). Turbulence in openchannel flows, IAHR Monograph Series, A.A. Balkema Publishers, Rotterdam, The Netherlands. Nezu, I. and M. Sanjou (2006). Numerical calculation of turbulence structure in depth-varying unsteady openchannel flows. Journal of Hydraulic Engineering, 132(7), 681–695. Nezu, I., A. Kadota and H. Nakagawa (1997). Turbulent structure in unsteady depth-varying open-channel flows. Journal of Hydraulic Engineering, 123 (9), 752–763. Nikuradse, J. (1933). Laws of flow in rough pipes, Translation in National Advisory Committee for aeronautics, technical memorandum 1292, NACA, Washington 1950, 62 p. Onitsuka, K. and I. Nezu (1999). Effect of unsteadiness on von Karman constant in unsteady open-channel flows. D1-Turbulent Channel Flows with Macro RoughnessVegetation, 28th Congress of IAHR, Graz, Austria, Conf. Proceedings. Powell, D. M., I. Reid and J. B. Laronne (2001). Evolution of bed load grain size distribution with increasing flow strength and the effect of flow duration on the caliber of bed load sediment yield in ephemeral gravel bed rivers. Water Resources Research, 37(5), 1463–1474. Qu, Z. (2002). Unsteady open-channel flow over a mobile bed. Ph.D.Thesis. No 2688, Ecole Polytechnique Fédérale (EPFL), Lausanne, Switzerland. Song, T. (1994). Velocity and turbulence distribution in nonuniform and unsteady open channel flow. PhD thesis, No. 1324, Ecole Polytechnique Fédérale (EPFL), Lausanne, Switzerland.

Song, T. and W. H. Graf (1996). Velocity and turbulence distribution in unsteady open channel flows. Journal of Hydraulic Engineering, 122(3), 141–154.

Tardu, S.F, G. Binder and R. F. Blackwelder. (1994).Turbulent channel flow with large-amplitude velocity oscillations. Journal of Fluid Mechanics, 267, 109–151. Tu, H. (1991).

Velocity distribution in unsteady flow over gravel beds. Ph.D. Thesis, No 911, Ecole Polytechnique Fédérale (EPFL), Lausanne, Switzerland. Tu, H. and W. H. Graf (1992). Velocity distribution in unsteady open channel flow over gravel beds. Journal of Hydroscience and Hydraulic Engineering, 10 (1), 11–25. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Non-linear optimization of a 1-D shallow water model and integration into

Simulink for operational use

L. Goffin, B.J. Dewals, S. Erpicum, M. Pirotton & P. Archambeau

HECE, University of Liege (ULg), Belgium

ABSTRACT: Dam operations leads to discharge changes in the downstream river. Evaluating the impact of

such changes can help to improve the management of these hydraulic structures. This can be achieved thanks to

a well calibrated 1-D model which offers a very low computational cost. The 1-D model accuracy is mainly a

function of the cross sections data as well as the calibration of a roughness coefficient. An original technique is

proposed to generate 1-D cross sections from existing 2-D steady simulations. A tuning factor can be added for

more flexibility. The Barr-Bathurst law is used for friction. The roughness size is also a parameter to calibrate.

The fitting process is achieved, in this paper, by a traditional space scanning method as well as by the simulated

annealing heuristic method. The practical case of the River Romanche, in France, is used for illustrating previous

developments. Finally, the integration of the model into Simulink is discussed.

1 INTRODUCTION

The Romanche, a river in the French Alps, is currently facing large changes. A new hydropower facility is being built in order to replace older power plants. For this project, as well as for many others around the World, it is important to be able to simulate quickly and efficiently the behavior of the river under a range of discharges consecutive to some human or natural inputs.

Precision, reactivity and operational integration are three important criteria for a simulation tool used for design and system control. Good precision can be reached easily thanks to traditional 2-D depth inte grated models. However, this kind of models does not allow fast computation for fine discretization unless GPU-based codes (Brodtkorb et al. 2012) are used. For traditional desktop computers, a simpler 1-D model can be very efficient. Even if it is less rich, it offers a good compromise between computation speed and results precision.

For many hydropower projects, fine 2-D simula tions (steady and unsteady) are firstly performed for determining the behavior of the river under given solic itations. Due to the high computational cost of such tools, they cannot be employed easily for further design steps, such as power production simulations or gate manipulations. Using a well calibrated and fast 1-D

model is ideal for this purpose.

In order to reach appreciable precision, equivalent

1-D cross sections were generated from previous cali

brated 2-D steady simulations. This approach leads to

richer information than local 1-D measures, based on

validated data. Following the implementation and the generation of 1-D cross sections, it was possible to optimize parameters. This calibration step was performed thanks to non-linear optimization tools implemented in the WOLF software. The calibrated model was then introduced into Simulink in order to be used inside already existing system modeling tools. This paper presents developments, implementations and results in the frame of the Romanche River. However, present developments can be easily extended to any other practical case. 2 1-D SHALLOW WATER MODEL WOLF1D uses shallow water equations in 1-D (SaintVenant equations) which express the conservation of mass and the conservation of momentum. These equations are expressed in a non-conservative form for a simpler approach of pressure terms: where [m 2] is the cross section area, Q [m 3 /s] the discharge, q L [m 2 /s] the lateral input discharge, u [m/s] the velocity in the direction of the discretization axis, β [-] a coefficient for unequal velocity distribution across the section, g [m/s 2] the gravity acceleration, Z [m] the position of the free surface, J [-] the friction

slope and θ [-] a coefficient which allows to take into

account or not the velocity of the lateral discharge.

As an initial condition, it is often necessary to stabi

lize a stationary solution. This can be done efficiently

by resolving the following equation (Kerger et al.

2011):

Equation (2) can stabilize a solution faster than by

solving the set of equations (1). Indeed, only one equa tion is solved and the number of nodes is adaptive with

pseudo-time τ [s]. At the beginning, only downstream

nodes can be considered.After that equation (2) results

in ∂/∂τ = 0, the number of nodes is increased down

stream to upstream and the system is stabilized again.

These operations are repeated until all the nodes are

treated.

Equations (1) and (2) are discretized thanks to a

finite volume approach.

2.1 Friction law

The friction slope can be written as

with the hydraulic radius $R_h =$ /χ [m], χ [m] being

the wet perimeter. The friction coefficient λ [-] is

defined by the Barr-Bathurst law (Machiels et al.

2011) which is well suited for macro roughness flows,

the case met in the Romanche. Let define k_s [m]

as the characteristic size of the bottom roughness

and $Re_* = R_h u/\nu$ [-], ν [m^2/s] being the kinematic

viscosity. For $k_s/h \leq 0.05$:

For $0.05 \leq k_s/h \leq 0.15$: For $k_s/R_h \geq 0.15$: Equation (5) ensures a smooth transition between Barr (4) and Bathurst (6) formulations. The BarrBathurst law offers the advantage to be explicit and physically based for a wide range of roughness size. 2.2 1-D cross sections Tabular values are needed to describe how the cross section area and the wet perimeter evolve with the water depth for each node. Usually, this kind of information comes from local measures. Since 2-D steady simulations were performed previously, they were used to determine 1-D cross sections. This approach leads to spatially richer information. Each node stretch is determined from the 2-D model thanks to a fast-marching algorithm (Sethian & Vladimirsky 2003). It solves the eikonal equation: where φ [m] is the curvilinear distance. Distances are determined from a reference section in the 2-D model. According to the distance desired between 1-D nodes, the stretch for each node can be extracted from the distribution of φ. Now that each node's influence zones are determined, tabular relations can be constructed. This process is based on a set of stationary simulations at various discharges. First, the bottom altitude z_b [m] of the node is determined from the minimum altitude in the influence zone. For each discharge, the wet cross-section [m^2] is which represents the ratio between the volume of water V [m^3] inside the influence zone and the 1-D node size [m]. For the wet perimeter, a first approach is to take the ratio between the horizontal wet area S [m^2] and the 1-D node size: The free surface altitude z_s [m] is computed as the mean of the free surface altitudes over the influence zone. The water height is given as $h = z_s - z_b$. For each stationary simulation a tuple {, χ , h} can be determined for every 1-D nodes. Even if the cross-sections construction method presented above allows to use the richness of a 2-D model, going 2-D to 1-D leads to a loss of information, especially transversally. In order to avoid this, we propose

a formulation for the wet perimeter, based on (9), that

includes a tuning parameter α:

with h_{max} [m] the maximum water height recorded with

the various discharges. When α = 0, the wet perimeter

is a function of the current water height h [m]. Indeed,

allowing a variation of the wet perimeter according to

the water height can reflect the uneven velocity dis

tribution across the section, and thus uneven friction

losses. The α coefficient should be fitted according to

reference results.

3 MODEL OPTIMISATION

The goal of using a 1-D model is, of course, to have

a significant gain in the execution time but it should be as close as possible to the 2-D model for chosen results. These results can include downstream hydro graphs or water height evolution for instance. This is achieved by calibrating some parameters. Even if the friction law is physically based, some freedom remains for determining the bottom roughness within a physi cal range. This parameter can be distributed along the river, in N zones, for a more sensitive fit, resulting in an N parameters optimization problem.

Assessing the “goodness-of-fit” between the 1-D model and the 2-D model, considered as the reference, is one of the key for succeeding in the calibration task. Some objective factors are presented in the next sub section.

Finally, the two optimization methods used in the frame of this paper are presented. Other methods are discussed for further developments.

3.1 Comparison factors

For the Romanche project, the goal was to repro duce accurately the temporal evolution of the discharge downstream.The objective function is the downstream hydrograph generated by the 2-D model. Comparing hydrographs can be done thanks to various factors.

The Nash-Sutcliffe efficiency coefficient (Jain & Sudheer 2008; Nash & Sutcliffe 1970) is expressed as with Q_i the reference discharge, $Q_{i,c}$ the computed

discharge with the 1-D model, Q_m the mean of the

reference discharges and N the number of discretiza

tion points of the hydrographs. This factor ranges from

0 to 1. NSE = 1 means that hydrographs fit perfectly.

NSE = 0 means that the 1-D model can only predict

the mean discharge. The index of agreement d (Legates & McCabe Jr. 1999; Willmott 1981) represents the ratio between the square error and the "potential error": Best results are obtained for d = 1. d is bounded between 0 and 1. The coefficient of determination R^2 expresses the proportion of the variance that the model is able to represents (Legates & McCabe Jr. 1999): $Q_{c,m}$ is the mean of the hydrograph computed by the model. R^2 values range from 0 to 1, higher values represents better agreement. 3.2 Optimization method Finding the best set of N parameters for calibrating a 1-D model is equivalent to an optimization problem. This kind of problems can be solved with different techniques: – By scanning the whole space of parameters – By evaluating the slope of a function and then determining the direction of evolution – Based on heuristic methods The first technique is the easiest to implement. However, a large number of combinations have to be tested if N is large and the refinement for each parameter is fine. Finding the optimum within bounds is certain but long computation times can be observed. For faster convergence, the Levenberg-Marquardt algorithm (Levenberg 1944; Marquardt 1963) can be used. It aims at minimizing the chi-squared error criterion by following a preferential evolution direction. The main drawback of this algorithm is that it can get stuck in a local minimum, according to the starting point chosen. In order to avoid this main drawback, the simulated annealing algorithm (Kirkpatrick et al. 1983; Metropolis et al. 1953), which is applied in various domains (Gilles et al. 2010), can be used. It is a heuristic method based on the same principle as annealing in metallurgy. At first, a high temperature T is chosen in order to assess many points in the parameters space. At each step, if a point offers a lower energy than the previous guess ($\Delta E < 0$) it is retained. Otherwise it is retained only if the probability $\exp(-\Delta E/T)$ is greater than a random probability. At each step, the temperature is cooled down according to a defined schedule, leading

Figure 1. Longitudinal profile of the Romanche River, from Livet to Gavet.

finally to a frozen state which represents the best solu tion that the algorithm was able to find. This method is sensitive to the initial temperature and the cooling schedule. It is very efficient for large number of param eters. It must be noticed that the final solution is not exact since the method is heuristic. In this paper, an improved algorithm is used (Corana et al. 1987; Goffe et al. 1994).

The first and third techniques will be applied in this paper.

Calibrating a model can be achieved on several reference situations. Each situation i leads to an effi ciency factor f_i (see subsection 3.1) which can be combined with factors from the other situations in order to use one of the optimization techniques pre sented above. The resulting combined factor f can be obtained thanks to the flowing relationship:

where K is the number of situations to consider and γ_i is a weighting factor for situation i.

3.3 Application to the Romanche

The studied part of the Romanche River stretches over 9.5 km, from the new Livet's Dam to Gavet. The longi tudinal profile of the river bed is given in Figure 1.Two zones can be identified: one on the first 500 meters and

a second one after this distance. The first zone has a

mean slope of 1.7 mm/m while the second zone has a

mean slope of 30 mm/m. These two different configu

rations suggest that different values of roughness size

can be observed. Moreover, cross sections are different

in each zone.

This observation leads to define two roughness

coefficients, k_{s1} and k_{s2} , and two coefficients for the

wet perimeter evaluation, α_1 and α_2 . Subscript 1 refers

to the upstream zone while subscript 2 refers to the

downstream zone. Table 1. Initial and maximum discharges for the 6 scenarios used for calibration. Q initial Q maximum Scenario m^3/s m^3/s 1 10 44 2 10 54 3 23 53 4 10 60.2 5 40 56.2 6 15 128 The 1-D domain is discretized with 50m-long meshes, resulting in a total of 191 nodes. The downstream boundary condition (BC) is an imposed water level at 436.8 m. In order to generate tabular relations that links the cross section area, the wet perimeter and the water depth, 13 2-D steady simulations were used. Discharges range from 4 m^3/s to 150 m^3/s. Thanks to the technique presented in section 2.2, a tabular relation was generated for every node. The 2-D model used is WOLF 2-D which implements shallow-water equations discretized in finite volumes (Erpicum et al. 2010).The studied Romanche section is discretized with 1×1 m^2 meshes, leading to 400,000 nodes. The friction law is the Barr-Bathurst law written in the two horizontal dimensions. The 2-D model was calibrated in comparison to on-site measures and previous numerical simulations. Six scenarios were used to calibrate the 1-D model. These scenarios are unsteady hydrographs injected at the upstream BC. The downstream hydrographs generated by the 2-D model are the reference data to which the 1-D model is compared. Information about hydrographs are given in Table 1 and Figure 4. Scenario 1 presents a steep increase in the discharge up to 42 m^3/s. Then, the discharge falls to 36 m^3/s to rise again to the maximum value. Then it decreases slowly to 40 m^3/s. The second scenario shows a first quick rise to 30.5 m^3/s and then a fall to the initial discharge. It rises again, this time irregularly, to the maximum discharge value. For the third hydrograph, the discharge increases almost regularly from the minimum to the maximum value. Scenario 4 shows an irregular increase from 10 to 60.2 m^3/s in 46 minutes. Then, it falls to 33.5 m^3/s and rises to 46.4 m^3/s. It finally decreases to a final discharge of 4 m^3/s. Scenario 5 does not present steep variations but globally higher discharges instead. Finally, scenario 6 starts from 15 m^3/s to reach a peak at 128 m^3/s and decreases to 93 m^3/s. For the optimization process, it was chosen to use the Nash-Sutcliffe efficiency coefficient (11) as function f to maximize. Each γ_i from formula (14) was taken to unity, giving the same weight to each scenario.

Figure 2. Nash-Sutcliffe efficiency coefficient as a function

of two roughness sizes (k_{s1} and k_{s2}). The best combination is

denoted by the cross mark.

The first optimization technique was used for the

calibration of the two roughness sizes (k_{s1} and k_{s2}).

The cross section wet perimeter was evaluated with

$\alpha = 0$ in equation (10). Values from 0.01 m to 1 m,

discretized with a 0.03 m step were used for both k_{s1}

and k_{s2}. This discretization results in 1156 couples of

parameters to test in the 1-D model. Each test is led

over 6 scenarios which extend together over 35.9 h.

This results in the simulation of 4.7 years of hydro

graphs. The best NSE coefficient found with this tech

nique was 0.98936 at $k_{s1} = 0.49$ m and $k_{s2} = 0.52$ m.

However Figure 2 shows that the upstream roughness

size has less influence, given the shorter hold.

As it can be seen in Figure 4, this calibration tech nique leads to a good agreement between reference hydrographs and results from the 1-D model. However, some problems can be noticed, especially for higher discharges, where the curves 1-D hydrographs gets away from the reference data. As presented in section 2.2, taking into account variation a modulating coeffi cient for the wet perimeter helps to better represent the uneven distribution of friction losses across sections of the river. Taking into account equation (10) brings two new parameters to calibrate, α_1 and α_2 , for zones 1 and 2 respectively. With the same number of parameters of k_{s1} and k_{s2} for α_1 and α_2 , the number of combinations of parameters would increase to 1,336,336. This can not be achieved within a reasonable time. This is why another optimization technique should be explored.

The simulated annealing method was used as the second technique. The number of model eval uation was limited to less than 2500 and the best NSE coefficient found was 0.99677 which is better than the previous one. This coefficient was obtained for k_{s1} = 0.4419 m, k_{s2} = 0.4060 m, α_1 = 0.05 and α_2 = 0.246. Resulting downstream hydrographs are given in Figure 4.

When comparing results obtained with both opti mization methods, it appears that the second technique

offer better agreement with reference data. Figure 3. Graphical representation of a Simulink model using a S-Function for a river application. 4 INTEGRATION INTO SIMULINK Integrating computational tools into already operational systems is a major concern for hydraulic facilities manager and designers. They need easy to use and easy to integrate components that can compute quickly the response of a river to solicitations. Simulink, which is part of the Matlab software, is a tool that allows to model and analyze dynamic systems. A river can be considered easily as a dynamic system. Indeed, following an input (e.g. an upstream discharge) it produces a corresponding response (e.g. water level elevation at a given point or downstream discharge). In Simulink, this is achieved by S-Functions. An S-Function can be user defined and coded in C or FORTRAN.The WOLF software being coded in the last FORTRAN standard, it can be used in Simulink.To do so, the WOLF software was compiled as a library. Two FORTRAN files are needed to interface with Simulink. A first one is provided by Matlab. The second has to follow some strict guidelines for Simulink to pass arguments to the routine. This file contains calls to the library in order to run WOLF 1-D model. Once compiled, a *.mex* file is generated and it can be distributed. For the Romanche, four inputs could be given to the routine: 1. an upstream discharge 2. a discharge from the hydro power plant, 150 upstream from the outflow BC 3. a downstream discharge 4. a water level at the downstream BC These 4 conditions are not needed simultaneously but can be combined in order to receive some output: the downstream discharge or the water level evolution at the downstream border. A graphical representation of the Simulink model is given in Figure 3. S-Functions can be very useful in the frame of industrial applications. Indeed, coupling such routine with the operational model of a dam can bring loads of information about how to manipulate properly, safely and efficiently the power plant and the gates for instance. A main drawback of this integration is the execution time. For instance, the overhead observed could reach

Figure 4. Comparison between the reference downstream hydrograph (from 2-D simulations) and downstream hydrographs

obtained from calibrated 1-D models, in regards to the upstream discharge. The first 1-D model (1) was calibrated with a scanning method only for two roughness sizes while the second 1-D model (2) was calibrated for two roughness coefficients

and two wet perimeter coefficient.

2 min when the discharge is updated every 60 s and 13 s

when the update is every 600 s (for a 24 h hydrograph).

5 CONCLUSION

Using a 1-D model instead of a 2-D model is relevant

when the computational time cost is a major concern.

In some practical cases, hours of flow should be sim

ulated in only a few seconds to deliver information

about, for example, discharge or water height at a given

cross section.

This paper focused on the generation of tabular rela

tions for a 1-D cross-sections and on the calibration of

a 1-D model. Both aspects were treated in regard to

reference 2-D simulations.

Concerning 1-D cross sections, 2-D steady simu

lations were used in order to compute, for a given discharge, a cross-section area, a wet perimeter and a water height. These three information are assembled into a tabular relation in order to define a 1-D crosssection. For the wet perimeter an original formulation was proposed for better calibration. The 1-D model calibration was done in regard to a 2-D model results.Two techniques were used.The first one focused on scanning a predefined space of coefficients.The second method, called simulated annealing, is based on random search. It was used to fit 4 parameters, including wet perimeters coefficients. Using these two techniques showed, on the Romanche case, that the simulated annealing can provide good results within fewer iterations than a traditional method. Adding coefficients for the computation of the wet perimeter appeared also to be very efficient. Finally, the paper showed how an efficient and well calibrated model can be integrated into Simulink. This integration is a powerful tool for operational users. In fact, being able to predict quickly and accurately the consequences of a manipulation can help in taking safe and efficient decisions.

Further developments should include a deeper anal ysis of the simulated annealing algorithm in order to make it more efficient in finding the best parameters for the calibration of a hydraulic model.

Brodtkorb,A.R., Sætra, M.L. &Altinakar, M., 2012. Efficient shallow water simulations on GPUs: Implementation, visualization, verification, and validation. Computers & Fluids, 55, pp. 1-12.

Corana, A. et al., 1987. Minimizing multimodal functions of continuous variables with the “simulated annealing” algorithm. ACM Transactions on Mathematical Software, 13(3), pp. 262-280.

Erpicum, S. et al., 2010. Dam break flow computation based on an efficient flux vector splitting. Journal of Computa tional and Applied Mathematics, 234(7), pp. 2143-2151.

Gilles, G., Habraken, A.M. & Duchêne, L., 2010. Material Parameter Identification of Cazacu’s Model for Ti6Al4V Using the Simulated Annealing Algorithm. In T. T. Publi cations, ed. Materials Science Forum. pp. 1125-1130.

Goffe, W.L., Ferrier, G.D. & Rogers, J., 1994. Global opti mization of statistical functions with simulated annealing.

Journal of Econometrics, 60(1), pp. 65–99.

Jain, S.K. & Sudheer, K.P., 2008. Fitting of hydrologic mod

els: a close look at the Nash-Sutcliffe index. Journal of

hydrologic engineering, 13(10), pp. 981–986. Kerger, F. et al., 2011. A fast universal solver for 1D continuous and discontinuous steady flows in rivers and pipes. International Journal for Numerical Methods in Fluids, 66(1), pp. 38–48. Kirkpatrick, S., Gelatt, C.D. & Vecchi, M.P., 1983. Optimization by simulated annealing. Science, 220(4598), pp. 671–680. Legates, D.R. & McCabe Jr., G.J., 1999. Evaluating the use of "goodness-of-fit" measures in hydrologic and hydroclimatic model validation. Water resources research, 35(1), pp. 233–241. Levenberg, K., 1944. A method for the solution of certain non-linear problems in least squares. The Quarterly of Applied Mathematics. Machiels, O. et al., 2011. Theoretical and numerical analysis of the influence of the bottom friction formulation in free surface flow modelling. Water SA, 37(2), pp. 221–228. Marquardt, D.W., 1963. An algorithm for least-squares estimation of nonlinear parameters. Journal of the Society for Industrial & Applied Mathematics, 11(2), pp. 431–441. Metropolis, N. et al., 1953. Equation of state calculations by fast computing machines.The journal of chemical physics, 21(6), pp. 1087–1092. Nash, J.E. & Sutcliffe, J.V., 1970. River flow forecasting through conceptual models part I—A discussion of principles. Journal of hydrology, 10(3), pp.282–290. Sethian, J.A. &Vladimirsky,A., 2003. Ordered upwind methods for static Hamilton-Jacobi equations: Theory and algorithms. SIAM Journal on Numerical Analysis, 41(1), pp. 325–363. Willmott, C.J., 1981. On the validation of models. Physical geography, 2(2), pp. 184–194. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Limitation of self-organization within a confined aquifer

M.C. Westhoff, S. Erpicum, P. Archambeau, M. Pirotton & B. Dewals

Hydraulics in Environmental and Civil Engineering (HECE), University of Liege (ULg), Liege, Belgium

E. Zehe

Karlsruhe Institute of Technology (KIT), Karlsruhe, Germany

ABSTRACT: Preferential flow paths are omnipresent in the subsurface, but very hard to observe or parameterize. Often they are even sub-grid processes requiring effective parametrization. In an ongoing study, Westhoff et al. (2016) show within a lab experiment that the steady state effective hydraulic conductance evolves (under certain circumstances) to the conductance that maximizes power by the flux through the confined aquifer. Here we explore why in one setup the effective conductance did obey this maximum power principle, while the same setup with slightly different boundary conditions did not lead to this. We suggest here, with a detailed numerical setup of the experiment, that the degrees of freedom to create a long enough preferential flow path was too limited in the latter case: a foam rubber layer, placed between the sand and the plate covering the sand (to pressurize the sand layer) prevented further development of the preferential flow path. While this preferential flow path was long enough to result in an effective conductance leading to maximum power in the first setup, it needed to be longer in the second setup.

1 INTRODUCTION

Soils act as filters receiving, storing and releasing water. It influences the water availability for plants as well as the amount of runoff draining into streams and rivers. While vegetation taps their water mainly from the soil matrix, most of the drained water flows through preferential flowpaths (Uhlenbrook 2006).These preferential flowpaths originate from cracking of soils (e.g.

by dry clay or freezing and thawing processes), animal burrows, old tree roots, dissolution of the porous media (in e.g. karst systems) or internal erosion processes. So, in order to predict streamflow generation or the residence time of e.g. pesticides in the soil system, it is important to properly consider these preferential flowpaths in your models.

The omnipresence of these flowpaths has been widely recognized. However, Beven and Germann (2013) argue that it should still get more attention than it currently gets: many models still consider the soil as a matrix whose properties are determined from small core samples analyzed in the lab, which is a too simple representation of reality. Why is this? First of all, Richards-based easy-to-use software packages, such as e.g. HYDRUS, are widely available, which limited the apparent need for new physical theories linking all kinds of flows .

Secondly, preferential flowpaths are extremely hard to observe directly. Available observations are mostly on the meter scale (or smaller) and mainly limited to vertical flowpaths (Noguchi et al. 1999, van Schaik et al. 2008). And although these observations give information on presence, network and size of the preferential flowpaths, it generally does not give information on the flow characteristics. And even when it does, upscaling these observations to the size of a hillslope is not straight forward. At this larger scale, tracer breakthrough curves can be analyzed to indirectly infer the effect of preferential flow paths. But to be comprehensive, such tracer tests should be repeated for different initial wetness states and different forcing rates (Wienhöfer et al. 2009). And thirdly, preferential flowpaths are often much smaller than the grid size of the model. It is thus a sub-grid phenomenon, requiring effective parameters to describe the integrated behaviour of the whole grid cell. However, to observe effective parameters, measurements are needed at the right scale (Savenije 2010), which are generally not available. Due to all this, current practice is to calibrate model parameters, resulting in an ill-posed problem with many parameter combinations revealing the same results. To bypass this calibration paradigm, self-organizing principles have been proposed stating that a natural system organizes itself in order to behave optimally with respect to an objective function. In literature, several objective function have been proposed such as minimized water stress (Rodriguez-Iturbe et al. 1999, Porporato et al. 2001, Caylor et al. 2009); maximum transpiration and minimal water and oxygen stress (Brolsma and Bierkens 2007), maximum net carbon profit (Schymanski et al. 2009) or more general thermodynamic optimality principles such as maximum entropy production (Kleidon and Schymanski 2008, Kleidon 2009, Porada et al. 2011, Wang and Bras 2011, del Jesus et al. 2012, Westhoff and Zehe 2013), maximum power (Kleidon et al. 2013, Kleidon and Renner 2013, Westhoff et al. 2014), minimum energy dissipation (Rinaldo et al. 1992, Rodriguez-Iturbe et al. 1992, Hergarten et al. 2014) or maximum free energy dissipation (Zehe et al. 2010, Zehe et al. 2013).

Although these studies reported success, major critiques were that these principles are not proven yet for hydrological systems and their limits of application are not defined clearly. This motivated the ongoing study of Westhoff et al. (2016) who aims to test within a lab experiment if the effective hydraulic conductivity of a saturated sand layer adapt to its boundary conditions in order to produce maximum power. A key aspect of this lab experiment was to design it such that the sand layer had enough degrees of freedom to adapt its effec tive hydraulic conductivity. They aimed to design the experiment such that the effective hydraulic conduc tivity adapts by piping; a backwards erosion process creating small pipes within the matrix. Although, the first promising results of Westhoff et al. (2016) indi cate that the maximum power principle indeed applies, some combinations of boundary conditions did not obey the maximum power principle.

Therefore, the overall aim of this paper is to explore why and when some of these combinations in bound ary conditions did not follow the maximum power principle. We do this by combining the observations of the lab experiment with high resolution 3D model ing to test a possible hypothesis that may explain this behaviour.

2 SHORT INTRODUCTION TO THE MAXIMUM POWER PRINCIPLE

Here, a very short description of the maximum power principle is given. For a more detailed explanation, the reader is referred to e.g. Kleidon and Renner (2013). The maximum power principle is derived from the first and second laws of thermodynamics. The first law states that energy is always conserved, the second states that entropy can only increase, implying a direc tion of change. From these two principles, the Carnot limit can be derived, stating that the maximum work or power (which is the rate of work over time) that can be subtracted from a fixed temperature gradient has a theoretical maximum at

where P_{Carnot} [W] is the theoretical maximum power, J_{hc} [W] is the heat flux from the hot to the cold reser voir and T_h [K] and T_c [K] are the temperature of the hot and cold reservoirs respectively. In the next step we consider the temperature of the hot and cold reser voirs as functions of i) an incoming heat flux in the hot

Figure 1. Sketch of experimental setup. Blue represents water and yellow the saturated sand. In the middle section, a foam rubber layer is compressed in between the top of the sand layer and cover.

reservoir, ii) outgoing heat fluxes which are a function of the temperature of the respective reservoirs and iii) the heat flux between the two reservoirs (J_{hc}). The last heat flux can be described as a thermal conductance times the temperature gradient. From this follows that there exists an effective thermal conductance that maximizes the power generated by the flux J_{hc}, due to the trade-off between the gradient and the flux. If power generated by water is considered (which is the case here) power is given by where ρ and g are the density [kg m^{-3}] and gravitational acceleration [m s^{-2}], Q_{12} and k_{12} are the discharge between the two reservoirs [m^3 s^{-1}] and the effective hydraulic conductance [m^2 s^{-1}] and h_1 and h_2 are the water levels [m] in reservoirs 1 and 2 respectively. Note that effective hydraulic conductivity can be retrieved by dividing the hydraulic conductance by the cross-sectional area and multiplying it by the length over which the water level difference takes place. 3 METHODS 3.1 Description of lab experiment The lab experiment is a hydrological translation of the model used by e.g. Lorenz et al. (2001) who optimized the atmospheric heat transport of different planetary systems with the maximum entropy production principle; a principle closely related with the maximum power principle. The setup consists of two reservoirs connected with a confined aquifer consisting of sand (Fig. 1). Only in one reservoir water is added at a constant rate (Q in) while both reservoirs drain freely. For both reservoirs the stage discharge relation is determined experimentally at the end of each run by draining both reservoirs while simultaneously determining the discharge from

the drain by measuring the increase of weight of a bucket hung under the drain. To keep the pressure on the sand layer, a thin layer of a few centimeter of foam rubber is compressed between the top of the sand layer and the cover of the confined aquifer. Within the foam rubber, nine pressure tubes are installed 10 cm apart of each other to observe the hydrostatic pressure at the top of the sand layer.

The idea is that once water is added in reser voir 1, a pressure difference builds up between the two reservoirs. If this pressure difference exceeds a certain threshold, piping is likely to start. This is a backward erosion process creating highly con ducting internal pipes within the soil matrix. Once these pipes evolve, the effective hydraulic conduc tance increases, increasing the discharge between the two reservoirs, which reduces the pressure difference between the two reservoirs. This feedback loop stops when the local pressure gradient drops below the ero sion threshold. Each single experiment was run until this steady state was reached. At this steady state, the effective hydraulic conductance is determined by $k_{12} = Q_{12}/(h_1 - h_2) = Q_2/(h_1 - h_2)$.

3.2 Numerical simulations

3.2.1 2-Reservoir model

To check if the observed effective hydraulic conduc tance corresponds with the conductance leading to a maximum in power, a numerical 2-Reservoir model is used (Fig. 2) in which the following equations are solved:

where Q_1 and Q_2 are the discharges draining reservoirs 1 and 2, respectively and are described by a power law function of the geometry: $Q_j = a_j (h_j - c_j)^{b_j}$, with a and b being constants to be determined empirically, c being the height of the drain opening above a reference level [m] and subscript j being the reservoir number. Q_{12} is the flow between the two reservoirs and is given as $Q_{12} = k_{12} (h_1 - h_2)$. A_1 and A_2 are the cross-sectional areas of both reservoirs.

In this model, the reservoirs 1 and 2 have the same cross-sectional area as the lab experiment. Q_{in} and the stage-discharge relations for Q_1 and Q_2 were deter mined empirically from the respective experiments.

For a single numerical run a certain value for k_{12} was assumed and the model was run until a steady state was reached (note that all parameters influence the steady state configuration, but k_{12} is the only unknown for a given experiment). Power was subsequently deter mined with Eq. (2). This was repeated for a whole range of k_{12} after which the k_{12} resulting in a max imum in power was determined. This optimal value for k_{12} was subsequently compared with the observed value. Figure 2. Sketch of numerical 2-Reservoir model.

3.2.2 3D model (WOLF 3D) The two reservoir model only describes a single effective conductance for the whole model domain. To test different hypotheses with respect to internal processes, a more detailed model is needed. For this we use WOLF-3D (Paulus et al. 2013). The model solves the Darcy equation in three directions with the finite difference method. Note that the Darcy equation uses a hydraulic conductivity [LT^{-1}] as opposed to effective hydraulic conductance [$L^2 T^{-1}$]. At the upstream and downstream sides of the model domain, constant head boundary conditions were applied. These heads were obtained iteratively, using the observed stagedischarge relations and the fluxes across the boundaries. In the discretization, grid sizes of 9 × 9 × 9 mm 3 were used. During the simulations a maximum of three different soil types were defined, each type with a different hydraulic conductivity: one representing the soil matrix, one representing the compressed foam rubber and one representing the pipe. Note that due to the Darcy-formulation, the flow in pipe elements is simulated as laminar flow as well. Pipe elements were simulated as a range of connecting grid cells with a relatively high hydraulic conductivity. To obtain a realistic shape of the pipe, we started by introducing a pipe at the most downstream border, consisting of only one grid cell and ran the model until steady state was reached. Subsequently, we defined the steepest hydraulic gradient between the most upstream grid cell of the pipe and the surrounding matrix grid cells. The matrix cell with the highest gradient is defined as the next connecting pipe grid cell. This was repeated until the pipe has a certain (arbitrary) length. This is done for a setting with and without the high conductive foam rubber layer on top of the sand. Power could be determined either in the same way as in the lab experiment – by only looking at the head differences and fluxes draining both reservoirs or by summing up the power produced by all fluxes between all grid cells. The difference between both methods is caused by rounding off errors and is negligible. To be able to compare the modelling results with the lab experiment, the simulated effective hydraulic conductance was determined in the same way as in the lab experiment ($k_{12} = Q_2 /(h_1 - h_2)$). In the optimization procedure, only the hydraulic conductivity of the pipe elements was varied, while the conductivities of the matrix and the foam rubber were kept constant.

Figure 3. Observed and fitted stage discharge relations of

both drains for the two experiments.

4 RESULTS

4.1 Experimental results

Here we present the results of two different lab exper

iments. Although we did not visually found a pipe, we

did see sand boiling up at the downstream boundary

of the sand layer during both experiments, indicat

ing that the local flow velocity was high enough to

transport sand particles. The inflow Q_{in} was simi

lar for both experiments (2.6 ± 0.05 and 2.85 ± 0.05

l/min, respectively) but the stage discharge relations

of the reservoir drains varied (Fig. 3). The first

experiment (Q_{in} = 2.6 ± 0.05 l/min) led to an observed value of k_{12} of $1.32 \cdot 10^{-5} \pm 1.67 \cdot 10^{-8}$ m^2/s. The optimized value of k_{12} with the 2-Reservoir model was $1.27 \cdot 10^{-5} \pm 1.67 \cdot 10^{-7}$ m^2/s (Fig. 4), which is very close to the observed value. This finding also corresponds with several other runs of Westhoff et al. (2016).

However, the second experiment we present in this paper did not lead to an observed k_{12} value corresponding to the one maximizing power. The stage discharge relation for this run was slightly steeper for both reservoirs due to a small change in the opening of both drains, meaning a higher discharge for the same water level. Also the incoming discharge was slightly higher ($Q_{in} = 2.85 \pm 0.05$ l/min). For this run the observed effective hydraulic conductance was $1.33 \cdot 10^{-5} \pm 5.0 \cdot 10^{-7}$ m^2/s, while the optimum effective conductance obtained with the 2-Reservoir model was $2.17 \cdot 10^{-5} \pm 1.67 \cdot 10^{-7}$ m^2/s (Fig. 4).

4.2 Developing and testing hypotheses

The results of experiment 1 are in line with the results of Westhoff et al. (2016) indicating that the maximum power principle can be used to predict the effective hydraulic conductivity of the sand layer. To explain Figure 4. Effective hydraulic conductance vs. power obtained with the two reservoir model for experiment 1 and experiment 2 and comparison with observed hydraulic conductance. why the results of experiment 2 did not correspond with these other findings we formulated the following hypothesis: the high conductance foam rubber layer causes the pipe to evolve to this layer after which it cannot evolve any further. While this pipe configuration was long enough to reach the state of maximum power within experiment 1, it was not long enough for experiment 2. To test this hypothesis, we used WOLF 3D to simulate groundwater flow in three directions. In the first setup we assumed no high conducting foam rubber layer. The evolved pipe is presented in Fig. (5a). Note that, in the determination of the pipe architecture as described in section 3.2.2, the pipe did evolve entirely to the upstream border, but we cut it off at an arbitrary length that could mimic the observed pressure levels of the first experiment reasonably well (Fig. 5c). In the second setup we did implement the high conductance foam rubber layer, leading to an evolved pipe connecting the foam rubber layer with the downstream boundary (Fig. 5b). Within the optimization procedure, we only varied the hydraulic conductivity of the pipe elements, while keeping the hydraulic conductivity of the matrix and the foam rubber (if present) constant at $2.2\cdot10^{-4}$ m/s and $9\cdot10^{-2}$ m/s respectively.The matrix hydraulic conductivity is estimated from the effective hydraulic conductivity right at the beginning of experiment 1 (when we expected no pipe has yet been developed or only limited), while the one for foam rubber was calibrated against observed pressure levels at the top of the sand bed during experiment 1. Note that both configurations could mimic the observed pressure levels at the top of the sand layer reasonably well, with a slightly better correspondence for the configuration with the foam rubber layer (5c).

Figure 5. 2-D slice of pipe layout in WOLF 3D for a) configuration without foam rubber and b) configuration with foam

rubber layer. c) observed and simulated pressure heads at the top of the confined aquifer for experiment 1.

Figure 6. a) Effective conductance (k_{12}) vs. power for the

second experiment determined with the 3D-model and b)

pipe conductance (k_{pipe}) vs. power for the second experiment

determined with the 3D-model.

4.3 Results of the 3D modelling

Optimizing the hydraulic conductivity of the pipe ele ment for experiment 1 gave the same result for both pipe settings when recalculated to effective hydraulic conductivity of the whole sand layer. These results were also the same as for the 2-Reservoir model (Fig. 4).

For the second experiment the results of both pipe configurations did differ. The first configuration (without foam rubber) did lead to a maximum in power in the same way as in the 2-Reservoir model (Fig. 4), while the second configuration did not lead to a maximum in power (Fig. 6a). Interestingly, the maximum simulated power of the configuration with the foam rubber is very close to the power belonging to the observed effective hydraulic conductivity ($P_{obs\,12} = 3.83 \cdot 10^{-6}$ and $P_{sim3D\,12} = 4.0 \cdot 10^{-6}$ m^4/s).This is another hint that this pipe configuration is at least very close to the real occurring one.

5 DISCUSSION

The simulations with WOLF 3D did not falsify our hypothesis that due to the foam rubber layer the pipe could not grow any further and it could therefore not result in a maximum in power. So we can state that the degrees of freedom to physically create a ‘long enough’ pipe were not sufficient for the second experiment. Why is this? The main reason is that the major flow path through the whole domain flows through the foam rubber. Because the pipe evolves from the downstream boundary in an upstream direction, it quickly reaches the foam rubber. Numerically, as described in section 3.2.2, the pipe could evolve much further. It would then follow the foam rubber from which it would only deviate at a much further upstream direction. However, this cannot happen in reality: no pipe can evolve in the foam rubber, since this is a fixed material. At the location where the numerical pipe would enter the sand layer again, this is physically not possible either.This is because a pipe is formed by internal erosion, meaning that sand particles are transported in a downstream direction. But because the downstream direction goes into the foam rubber, the particles are blocked and cannot go anywhere. So no pipe can evolve there either. Of course, this whole reasoning also applies to the first experiment, which did obey the maximum power principle, while the only differences are the stage discharge relations of both reservoirs. So what explains this? To answer this we have to take into account that in any cross-section perpendicular to the mean flow direction the flux should be the same. Thus also in the part where no pipe has been developed. Over these cross-sections the potential gradient should be steeper than cross-sections where a pipe is present. If we now optimize the pipe conductance (k_{pipe}) for the

second experiment with WOLF 3D and subsequently determine the effective conductance, no maximum in power exists (Fig. 6a), even over a very wide range of pipe conductances (Fig. 6b). This shows that with this pipe configuration, the effective conductance hardly changes anymore at very large pipe conductances. To understand this better, let's assume an extreme case of a pipe consisting of only one grid cell (at the downstream boundary). Even if the conductance of this single grid cell goes towards infinity, it hardly affects the effec tive conductance. Note that the critical pipe length depends on the boundary conditions. In this paper only the stage discharge relations are different between the two experiments, but it can be expected that it also depends on the incoming discharge (Q_{in}).

Another point worth to mention is the fact that the observed effective conductance of the second experiment was only marginally larger than the first

($1.33 \cdot 10^{-5}$ vs. $1.32 \cdot 10^{-5}$ m^2/s), suggesting that with this pipe configuration the observed conductance is at it's physical maximum; e.g. a larger pipe may collapse due to the small cohesion of the used sand. However, this should be investigated further.

As a last remark, we would like to state that the modelled or calibrated hydraulic conductivities of the pipe and foam rubber depend on their respective cross sections. In our simulations we gave the foam rubber a thickness of 1 grid cell of 9 mm and the pipe a cross-section of one grid cell of 9 × 9 mm. Making this smaller or larger would result in respectively, a higher or a lower calibrated hydraulic conductivity. However, this would not influence all other results including the effective conductance k_{12}.

6 SUMMARY AND CONCLUDING REMARKS

In this paper we investigated why in the first configura tion of boundary conditions the effective conductance of the confined aquifer did obey the maximum power principle, while the second configuration did not. We hypothesized that this is because for the second config uration the degrees of freedom to form a pipe through the sand layer were too limited.

We were able to confirm this hypothesis with a high

resolution WOLF 3D model in which the pipe configu
ration could be modelled explicitly. However, we were not able to confront the modelled internal structure of the aquifer with observations, since the pipe did not persist once the experimental setup was opened and the sand excavated. Therefore, the proposed hypoth
esis is not proven in the sense that it is the absolute truth. Instead we merely showed that it is a reasonable hypothesis which we could not reject with our detailed 3D groundwater model.

We are therefore open for other hypotheses explain
ing the observed discrepancy between observed and optimized effective hydraulic conductance for the sec
ond experiment, while it should also be able to show that or explain why this discrepancy is absent in the

Rodriguez-Iturbe, I., P. D'Odorico, & L. Porporato, A. and Ridolfi (1999). On the spatial and temporal links between vegetation, climate, and soil moisture. Water Resour. Res. 35(12), 3709–3722.

Rodriguez-Iturbe, I., A. Rinaldo, R. Rigon, R. L. Bras, E. Ijjasz-Vasquez, & A. Marani (1992). Fractal structures as least energy patterns:The case of river networks. Geophys. Res. Lett. 19(9), 889–892.

Savenije, H. H. G. (2010). HESS Opinions "Topogra
phy driven conceptual modelling (FLEX-topo)". Hydrol.

Earth Syst. Sci. 14(12), 2681–2692. http://www.hydrol

earth-syst-sci.net/14/2681/2010/.

Schymanski, S. J., M. Sivapalan, M. L. Roderick, L. B.

Hutley, & J. Beringer (2009). An optimality-based model

of the dynamic feedbacks between natural vegetation and

the water balance. Water Resour. Res. 45(1), W01412.

Uhlenbrook, S. (2006). Catchment hydrology – a science

in which all processes are preferential. Hydrological

Processes 20(16), 3581–3585.

van Schaik, N. L. M. B., S. Schnabel, & V. G. Jetten (2008).

The influence of preferential flow on hillslope hydrology

in a semi-arid watershed (in the spanish dehesas). Hydrol.

Processes 22(18), 3844–3855.

Wang, J. & R. L. Bras (2011). A model of evapotranspira

tion based on the theory of maximum entropy production.

Water Resour. Res. 47(3), W03521. Westhoff, M., S. Erpicum, P. Archambeau, M. Pirotton, E. Zehe, & B. Dewals (2016). Experimental proof that the effective soil hydraulic conductivity can be predicted with the maximum power principle. in preparation. Westhoff, M. C. & E. Zehe (2013). Maximum entropy production: can it be used to constrain conceptual hydrological models? Hydrol. Earth Syst. Sci. 17(8), 3141–3157. Westhoff, M. C., E. Zehe, & S. J. Schymanski (2014). Importance of temporal variability for hydrological predictions based on the maximum entropy production principle. Geophys. Res. Lett. 41(1), 67–73. Wienhöfer, J., K. Germer, F. Lindenmaier, A. Färber, & E. Zehe (2009). Applied tracers for the observation of subsurface stormflow at the hillslope scale. Hydrol. Earth Syst. Sci. 13(7), 1145–1161. Zehe, E., T. Blume, & G. Blöschl (2010). The principle of 'maximum energy dissipation': a novel thermodynamic perspective on rapid water flow in connected soil structures. Phil. Trans. R. Soc. B 365(1545), 1377–1386. Zehe, E., U. Ehret, T. Blume,

A. Kleidon, U. Scherer, & M. Westhoff (2013). A thermodynamic approach to link self-organization, preferential flow and rainfall–runoff behaviour. Hydrol. Earth Syst. Sci. 17(11), 4297–4322. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Preliminary experiments on the evolution of river dunes

G. Oliveto & M.C. Marino

University of Basilicata, School of Engineering, Potenza, Italy

ABSTRACT: Dunes are common large-scale bedforms in sand-bed rivers that can affect significantly hydraulic roughness and water levels. This paper is a part of ongoing investigations by the authors on the evolution of river dunes. Here the spatial and temporal development of solitary sand dunes is investigated experimentally. Experiments were carried out at University of Basilicata, Italy, in a 1 m wide and 20 m long rectangular channel. A nearly-uniform sand with median grain size d_{50} = 1.7 mm was used as mobile bed. The working section was 16 m long. Each run was performed in two phases. Phase 1 aimed to generate the typical shape of the dune starting from an arc-shaped sand bar, of the same material as the mobile bed, set in a cross section perpendicular to the longitudinal channel axis. Phase 2 aimed to promote the downstream propagation of dunes generated in Phase 1 by either lowering the tailwater level or increasing the discharge. Based on the experimental data new insights are provided on the equilibrium morphology and the rate of migration of solitary sand dunes.

1 INTRODUCTION

The hydraulic roughness of rivers depends greatly on the bedform configuration. Dunes are common bed forms that typically develop in beds with sediment sizes from medium sand to gravel.They tend to migrate downstream and are out of phase with the water surface waves. Their longitudinal profiles are often asymmet rical with fairly mild stoss side slopes (around 5 ∘) and steeper lee side slopes close to the angle of repose of the bed material (Julien 2010).

An empirical study on dune features was made among others by Van Rijn (1984), who suggested some functional relationships for height, length and steepness of dunes. The height of river dunes is in the order of 10 to 30% of the water depth and the length (i.e. distance between two consecutive crests) is about 10 times their height. Further important contri butions in this regard are those by Kennedy & Odgaard (1991) and Julien & Klaassen (1995). Kennedy & Odgaard (1991) proposed an analytical model for dune height that is concerned primarily with megadunes. Julien & Klaassen (1995) investigated the geometry of lower regime bedforms in several large sand-bed rivers during average and flood conditions.

Solitary dunes can also be observed in nature. How ever, the comprehension of their development is of great importance in the understanding of the evolu tion of composite dunes. An earlier study was made by

Engelund (1971). He proved using a hydraulic model

that a solitary travelling sand hummock could exist at

Froude number smaller than 0.6 and within a certain

narrow range of sediment transport conditions. In his

report,Termes (1984) theoretically and experimentally studied the water movement and the sediment transport along a solitary dune. In particular, he found that relaxation is not the only mechanism to explain the length of sand dune. Gisonni & Hager (2009) investigated experimentally the main hydraulic features of solitary dunes. The dune migration was surveyed allowing to estimate the evolution of dune length and height, together with dune celerity. An approach to predict the dune celerity based on the classical bed-load formula by Meyer-Peter and Müller was proposed. Finally, special experiments in a narrow water flow tube and using glass beads as sediment were carried out by Groh et al. (2009) to investigate the spatio-temporal behavior of barchan dunes.The development towards a shape attractor is shown, though four different starting configurations were considered. Although substantial advances have been made in the last years, there are still several key research questions that require attention to yield a more exhaustive understanding of natural alluvial dunes (Best 2005). Based on experimental investigations, this paper would like to provide some outcomes on the equilibrium morphology and the celerity of solitary sand dunes. The experimental work by Gisonni & Hager (2009) is taken as the starting point for this study. The experimental data also provide new information useful for validation of advanced numerical models. 2 EXPERIMENTS Experiments were performed in a 1 m wide, 1 m deep, and 20 m long rectangular straight channel at the University of Basilicata, Italy.

Table 1. Main test conditions and dune characteristics for

runs of Phase 1.

Tests	Q [m 3 /s]	h o [m]	o [m]	o [m]	t[h]
A1	0.065	0.20	0.070	0.380	1.5
B1	0.070	0.20	0.070	0.380	1.5
C1	0.075	0.20	0.070	0.380	1.5

An almost uniform sand with density ρ_s = 2650 kg/m^3, median grain size d_{50} = 1.7 mm, and sediment gradation $\sigma = (d_{84}/d_{16})^{1/2}$ = 1.5 was used. The mobile bed was 16 m long to guarantee fully developed approach flow and make available enough bed length for dune propagation. The undisturbed bed surface was flat and horizontal. All the experiments were performed under steady flow conditions and clear-water scour regime.

The water discharge was measured to an accuracy of ±3% by an orifice plate installed in the circuit. The water surface was measured by a conventional point gauge with accuracy to the nearest millimetre and the sediment surface was measured to an accuracy of the order of the grain size by a shoe gauge with a 4 mm by 2 mm wide horizontal plate at its base. Detailed measurements of the bed morphology (typically around 1000 bed-level data) were taken during and at the end of each run. The approach flow depth was controlled by an adjustable sharp-crested weir located at the end of the channel.

Once the bed was carefully levelled, the channel was slowly filled to avoid any sediment movement by submerging the working section with the weir.

Some refinements to the starting dune configuration were also made under quitewater conditions The water discharge was gradually increased to the preselected value. The experiment started when the approach flow depth was set to the preselected value by lowering the weir.

Runs were performed in two phases according to the experimental method suggested by Gisonni & Hager (2009). Phase 1 aimed to generate steady-state dune shapes starting from an initial arc-shaped dune, of the same material as the mobile bed, with the axis per pendicular to the longitudinal channel axis. Phase 2 aimed to promote the dune propagation starting from the natural-like dune generated in Phase 1 by low ering the tailwater level, but keeping constant the discharge. Phase 2 would then simulate the evolution of dune morphology under unsteady flow conditions when flow depth decreases but the discharge remains constant.

Tables 1 and 2 give the main conditions for each run with Q = discharge, h o = approach flow depth, o and o = dune height and dune length at run starting, and t = time from run starting. and will be better defined in the next section.

Figure 1 shows the development of the starting arc shaped dune towards a more natural dune form in case of run A1. A diagram with the axial bed profile is Table 2. Main test conditions and dune characteristics for runs of Phase 2. Tests Q [m 3 /s] h o [m] o [m] o [m] t[h] A2 0.065 0.17 0.057 0.540 22.5 B2 0.070 0.17 0.049 0.560 22.5 C2 0.075 0.17 0.043 0.640 22.5 Figure 1. Evolution during run A1 of the arc-shaped sand dune towards its steady-state shape. Photographs correspond to times t from run starting (a) t = 0 h, (b) t = 0.1 h, and (c) t = 1.5 h. z is the bed elevation (in cm) and x is the axial (longitudinal) coordinate (in cm). Gray symbols represent the starting dune configuration. associated with each photograph to better appreciate the dune geometrical characteristics. During the time the initial heap becomes lower and longer. The starting symmetrical arc-shaped longitudinal profile evolves into a markedly asymmetrical profile with a gently inclined stoss side and a steeper lee face. In this run the slope of the stoss side was found around 3 ∘ while that of the lee side around 50 ∘ (i.e. somewhat larger compared to the angle of repose of the bed material) at the equilibrium phase. Figure 2 shows the dune morphology evolution during run B2 starting from the dune shape attained at the end of run B1. In comparison to the previously described run A1, the initial dune becomes further lower and longer due to the increasing approach Froude number. The slope of both the stoss and lee sides is definitely smaller and also the time to achieve the new equilibrium state is definitely longer and around 23 hours. 3 BASIC OBSERVATIONS Main experimental observations are discussed in this section. Figure 3 provides a definition sketch for a

Figure 2. Evolution of dune morphology during run B2.

Photographs correspond to times t from run starting (a)

t = 0 h, (b) t = 0.75 h, (c) t = 6 h, and (d) t = 22.5 h. z is

the bed elevation (in cm) and x is the axial (longitudinal)

coordinate (in cm). Gray symbols represent the starting dune

configuration.

Figure 3. Definition sketch for a solitary dune.

solitary dune with �= dune height, �= dune length,

α = stoss side angle, and β = lee side angle.

Here the dune length, �, is defined as the distance from the beginning of the stoss side to the end of the lee side while the dune height, ▯, is the vertical dis tance between the crest and the undisturbed bed level (as, for example, in the paper by Tjerry & Fredsøe 2005). Tables 3 and 4 give the main geometrical char acteristics for the central slice of the observed dunes at the end of each experiment. Subscript “e” stands for equilibrium state and is the aspect ratio of length to height.

The approach Froude number F o for runs of Phase 1 ranges from 0.23 to 0.27. For these runs substantial dunes formed at the equilibrium stage with aspect ratio up to 15. In case of runs of Phase 2, F o ranges from 0.27 to 0.31, but dunes evolve towards rather elon

gated shape with up to 205. Then, the morphology

Table 3. Main test conditions and dune characteristics for runs of Phase 1 at the equilibrium state.

Tests	e [m]	e [m]	α e [∘]	β e [∘]	e [–]
A1	0.057	0.540	2.5	50.2	9.6
B1	0.049	0.560	1.5	35.7	11.4
C1	0.043	0.640	1.4	17.0	15.0

Table 4. Main test conditions and dune characteristics for runs of Phase 2 at the equilibrium state.

Tests	e [m]	e [m]	α e [∘]	β e [∘]	e [–]
A2	0.015	1.740	0.3	20.1	116.0
B2	0.017	2.040	0.2	11.3	120.0
C2	0.017	3.380	0.2	6.2	204.8

Table 5. Observed averaged values of c D for runs of Phase 2. Values are in [m/s] and multiplied by 10 5 .

Tests	(▯t) 1	(▯t) 2	(▯t) 3	(▯t) 4	(▯t) 5
A2	90.00	11.67	6.11	3.48	0.43
B2	83.33	16.67	5.56	2.52	0.82
C2	76.67	13.33	7.78	4.44	2.73

of dunes is clearly dependent on F o with a more than linear dependence on F o . Also α and β would appear dependent on F o with values of α lower than 5 ∘ and values of β which are not strictly corresponding to the angle of repose of bed sediment. According to Oliveto & Hager (2014), the celerity of a solitary dune, c D , can be defined as the speed of downstream movement of the dune crest (or the dune front). c D can be then estimated as the ratio of the difference of two consecutive observations of over the time interval elapsed between them. Table 5 provides the observed averaged values of c_D during runs of Phase 2. Here c_D was estimated considering the movement of the dune front. Subscripts 1 to 5 refer to the five consecutive time intervals $(\Delta t)_1$ to $(\Delta t)_5$ in which the run duration was divided after starting. In particular, $(\Delta t)_1$ = 300 s, $(\Delta t)_2$ = 600 s, $(\Delta t)_3$ = 1800 s, $(\Delta t)_4$ = 13500 s, and $(\Delta t)_5$ = 64800 s. From Table 5 it can be found that the weighted average values of c_D are $1.48\cdot10^{-5}$, $1.63\cdot10^{-5}$, and $3.48\cdot10^{-5}$ m/s for runs A2, B2, and C2, respectively. As expected, it can be concluded that for a given run c_D exponentially decreases over time and increases as F_o increases. 4 COMPARISON WITH 2D SIMULATIONS Gisonni & Hager (2009) presented a novel (onedimensional) approach to predict the main dune

Figure 4. Comparison between 2D simulated (bottom) and

observed (top) bed elevations at the end of run B2. Bed levels

are in meters (initial horizontal bed level = 0.675 m).

parameters based on the classical bed load theory

of Meyer-Peter and Müller. They proposed straight

forward relationships able to predict the height and

celerity of solitary dunes satisfactorily. In order to

enhance their results the CCHE2D model was applied

to simulate the evolution of solitary dunes. Here,

the analysis was restricted to the experiments of

Phase 1.

CCHE2D (Zhang 2005) is a depth-integrated 2D

hydrodynamic model for unsteady turbulent open

channel flow and sediment transport simulations

developed at the University of Mississippi. A variant

of the finite element method is applied to discretize the governing equations and the time marching technique is used for temporal variations.

The computational meshes were generated using the software CCHE2D Mesh Generator. Meshes with high orthogonality, aspect-ratio and smoothness were achieved.A simulation domain 20 m long and 1 m wide was chosen. The triangulation interpolation was used as interpolation algorithm.

The depth-integrated mixing length model was adopted as eddy viscosity model. Moreover, the Manning's n bed roughness coefficient was estimated as $1/n = 6.75 g^{1/2} d_{50}^{-1/6}$ according to Strickler. The roughness height was assumed zero (no slip condition) at smooth glass walls.

In regard to sediment transport, two layers were defined for the bed material sorting calculations. The first layer (mixing layer) thickness was assumed equal to 0.05 m while the second layer was 0.60 m consistently with laboratory conditions. The average saltation length was assumed equal to 7.3h, with h flow depth, that is the theoretical value for the average length of dunes (van Rijn 1984). Four different transport capacity formulas are available to the user. In this study the modified Ackers and White's formula (Zhang 2005) was considered.

A constant discharge was imposed at the inlet

boundary and a constant water level at the outlet. A

time step of 0.1 s was considered for the (unsteady)

sediment transport simulation.

Figure 4 shows the comparison between the com

puted and observed bed elevations at the end of run

B2. Bed patterns look reasonably similar. Both, the

computed and measured topographies show as the cen

tral slices of the initial heap migrated with the fastest

velocities. In contrast, the sidemost slices advanced

downstream with the smallest velocities. In this man

ner, a parabolic shape of the dune front was formed. Figure 5. Comparison between (top) simulated (▮) and measured (▮) axial water surface profiles at the end of run B1, (middle) simulated (•) and measured (∘) axial bed profiles at the end of run B1, and (bottom) simulated (•) and measured (∘) axial bed profiles at the end of run B2. z is the water/bed elevation (in cm) and x is the axial (longitudinal) coordinate (in cm). Gray symbols represent the starting dune configuration. In coherence with that, the central slices exhibited the smallest dune heights while the sidemost ones the largest. In particular, Figure 5 compares simulated axial bed profiles with laboratory measurements at the end of runs B1 and B2. The comparison between simulated and observed axial water surface profiles at the end of run B1 is also provided. Both simulated and measured water profiles are out of phase with the bed profile for a length in concurrence with the dune elongation. Incidentally, there were no significant differences between simulated water profiles when applying either mixing length, or parabolic, or k-ε scheme as eddy viscosity models.

In the case of run B1, the simulated dune has height e

of 0.049 m equal to the observed value of e and width

e of 0.40 m somewhat smaller than the observed e of 0.56 m. Similar results were found for runs A1 and C1. Overall, it can be concluded that for runs of Phase 1 the 2D model predicts the dune depth at the equilib rium stage satisfactorily, however it does not explain the observed elongated dune form. This could be due to the lack of ability of the 2D model to simulate the flow separation zone at the dune lee side. Incidentally, in regard to the simulated results for run B1 it should be remarked that the sediment eroded from the ini tial arc-shaped dune is mainly transported downstream and deposited as a very thin layer along the dune lee side and the horizontal bed. Only a smaller part of it is deposited along the dune-stoss side. Similar results were found in the case of run B2, though the simu lated dune height and length are practically the same as experimentally observed. Conversely, the simulated dune shape appears rather different from the observed one (that's mainly along the channel axis) with the length of the lee side around 1.4 m (observed value around 0.25 m) and the length of the stoss side around 0.5 m (observed value around 1.6 m). Similar results were found for runs A2 and C2.

5 CONCLUSIONS

Laboratory experiments on the evolution of solitary sand dunes were carried out at the Hydraulic Engi neering Laboratory, University of Basilicata, Italy. Experiments were performed under steady flow conditions and clear-water scour regime. They were organized into two phases: Phase 1 intended to generate steady-state dune shapes starting from an arc-shaped dune; Phase 2 aimed to explore dune migration starting from the natural-like dunes gener ated in Phase 1. Runs lasted from 1.5 to 22.5 hours until quasi-equilibrium conditions were attained. The approach Froude number F_o varied from 0.23 to 0.31 and the relative submergence h_o/d_{50} from 106 to 118. The main results can be summarized as follows: (i) with regard to runs of Phase 1 the starting symmetrical arc-shaped profile evolved into an asymmetrical pro file with a gently inclined stoss side, which slope was somewhat smaller than 5° and a steeper lee face, which slope was found dependent on the approach Froude number and not strictly corresponding to the angle of repose of the bed sediment; (ii) developed dunes from runs of Phase 2 were much lower and longer compared to those of Phase 1 with aspect ratio up to 205. The slopes of the stoss and lee sides were defi nitely lower than 5° and the angle of repose of the bed material, respectively. Dunes migrated downstream with decrement of migration rate over time. Migration rates showed, on average, a decrease from 83.3·10 −5 to 1.3·10 −5 m/s during 22.5 hours; (iii) interestingly, the experimental observations on dune geometry appear in agreement with real conditions. In the present study,

Termes, A.P.P. 1984. Water-movement over a horizontal bed and solitary sand-dune. Rep. R/1984/H/8: Delft University of Technology, Department of Civil Engineering.

Tjerry, S. & Fredsøe, J. 2005. Calculation of dune morphology. Journal of Geophysical Research 110(4): Article number F04013. van Rijn, L.C. 1984. Sediment transport, part III: Bed forms and alluvial roughness. Journal of Hydraulic Engineering 110(12): 1733–1754. Zhang, Y. 2005. CCHE2D-GUI – Graphical User Interface for the CCHE2D Model User's Manual – Version 2.2. Technical Report No. NCCHE-TR-2005-03, School of Engineering, The University of Mississippi, USA. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

A conceptual sediment transport simulator based on the particle size distribution

B.T. Woldegiorgis, W. Bauwens & M. Chen

Department of hydrology and hydraulic engineering, Vrije Universiteit Brussel, Belgium

F. Pereira

Flanders Hydraulics Research (Waterbouwkundig Laboratorium), Belgium

A. van Griensven

Department of hydrology and hydraulic engineering, Vrije Universiteit Brussel, Belgium

Chair group of hydrology and water resources, UNESCO-IHE Institute for Water Education, The Netherlands

ABSTRACT: Sediment transport is a very important process affecting the transport and retention of pollutants

that easily adsorb to suspended sediment. The fine sediment plays a dominant role and needs to be properly

represented. However, the conventional conceptual sediment transport models do not account for distributions of

particle sizes and the differences in behaviour of the different fractions. Our paper presents an analytic solution

for this problem using the log-normal probability density function to represent the particle size distribution of the

sediment. Sedimentation and resuspension processes are calculated for the particle size distributions by using

Hijulstrom diagram for incipient motion and deposition and integrating with Velikanov's energy of sediment

carrying capacity. An application on the Zenne River (Belgium) shows that our conceptual model provides a

better representation of the high concentrations and the range of concentrations, as compared to a conventional

conceptual sediment transport modelling.

1 INTRODUCTION

Sediment transport plays an important role in char

acterizing the water quality behaviour of rivers and

sewers. Several pollutants are transported attached to

the suspended sediments (Jamieson et al., 2005; Lewis

et al., 2013; Miller et al., 1982; Ouattara et al., 2011;

Viney et al., 2000). In this regard, sedimentation and

resuspension processes control the transport and reten

tion of contaminants that show affinity to sediments

(Ani et al., 2011; Corbett, 2010; Crabill et al., 1999; Dhi, 2007; Ongley, 1996). Therefore, estimating the amount of sediment in the water is essential to model the dynamics of attached pollutants.

The sediment-bound pollutant transport is dom inated by the suspended sediments (Ongley, 1996; Sartor, 1972). The latter indeed contain much more of the fine particles (as compared to the bed load) as the process of surface erosion tends to be selective towards smaller particles (Asselman, 2000; Kumar & Rastogi, 1987).Therefore, it is important to distinguish the fine sediments from the total suspended sediment (Church & Krishnappan, 1998).

Detailed, hydrodynamic, sediment transport mod els are able to represent the sediment particles in to different size classes and perform the computation on each fraction (Dhi, 2007; Shrestha, 2013;Young et al., 1989). This approach increases the computation time and cost because the update of each sediment size class has to be maintained at every computation time step and every reach. On the other hand, surrogate models are ideal for fast computations but they –usuallydo not make a distinction between the proportions of the different particle sizes of sediments. Instead, they are based on the stream carrying capacity to determine the resuspension or deposition processes (Viney et al., 2000; Williams, 1980). Consequently, they lack essential detail for simulating the transport of pollutants attached to the sediments. In order to make maximum use of the computational efficiency and simulate the sediment-attached contaminants, we developed a surrogate sediment transport model that represents the particle size distribution by probability density functions and account for the deposition and resuspension phenomena. Our model calculates the sediment mass and the particle size distribution –i.e. the mean and standard deviation of the distribution-, using a simplified concept for the transport and resuspension processes. We determined the critical particle size of deposition and resuspension separately based on the Hjulström diagram. The

critical conditions of resuspension and deposition of

the new conceptual model are applied along with

Velikanov's energy , to calculate the sediment carrying

capacity (Velikanov, 1954).

The performance of the new model was tested on

the case study of the Zenne river in Belgium. It was

compared with a simple continuous stirred tank reac

tors (CSTR)based sediment model, combined with

Velikanov's energy.

2 METHODOLOGY

2.1 The study area

A reach of 41 km of the Zenne River –a lowland river in

Belgiumwas used as a case study for the evaluation of

a new conceptual sediment transport simulator. All the

tributary rivers and the inlet from the upper reach were

considered as point source boundaries. As the river is

subject to tidal backwater, the station just upstream of

the tidal influence (at Eppegem) was taken as the most

downstream point of the model.

Based on the cumulative frequency curves of the

sediment particles (Shrestha, 2013), we identified four

categories of sediment sources with specific character istics. Studies show that grain size distributions vary seasonally (Buscombe et al., 2014). No data was available regarding the temporal variability of grain size distributions for the Zenne stations. Therefore, we assumed the sediment characteristics do not change with time.

Sediment concentrations are measured measured by VMM(Flemish environment agency) on an average time interval of one month.

2.2 Theories of sediment transport

Bertrand-Krajewski (2006) concluded in his review paper that water quality models often ignore the cohesive nature of sediments for simplicity. Guo et al. (2012), however, showed that Stokes' law is applicable only to the sedimentation rate of sand particles and not to cohesive sediment. Bertrand-Krajewski (2006) stated that fine sediments are cohesive especially in the presence of bacteria released from effluents of waste water treatment plants and CSOs that glue sediment particles. The cohesive properties of sediments become dominant when the clay fraction is larger than 10%(van Rijn, 1993). The measurement data of the particle size distribution of the sediments in the river Zenne(Shrestha, 2013), however, show that the clay fraction is generally below 10%. Moreover, analysis of historical hydraulic simulation results by Shrestha

(2013), show that turbulent flow characteristics prevail

in the Zenne river, thus it hampers both floc formation

and growth in suspension mode(Church & Krishnap

pan, 1998). Therefore, we adopted a non-cohesive

deposition of coarse sediments. In order to account

for aggregation of fine sediments after depositing

to the bed(Jain and Kothyari, 2010), we adopted a non-cohesive theory of sediment resuspension. The well-known Shields curve (Shields, 1936) used in sediment transport relates the dimensionless shear stress to the particle Reynolds number (Equation 2.1). where ϑ is the dimensionless shear stress; R ∗ is the particle Reynolds number; τ is the bed shear stress [N/m2]; ρ s andρ are particle density and water density [kg/m 3], respectively; u ∗ is the shear velocity [m/s]; υ is the kinematic viscosity of water [m 2 /s]; g is acceleration due to gravity [ms 2]; R is the hydraulic radius [m]; S is the reach slope[m/m]. The implicit nature of Shields criteria makes Shields diagram difficult to interpret(Paphitis, 2001) and hence complicates its implementation in conceptual models. A more recent algebraic (Equation 2.2) developed by Soulsby & Whitehouse (1997) is a good representation of Shields criteria (Miedema, 2010; Shrestha, 2013). However, it is only applicable to non-cohesive sediments (Miedema, 2010). Besides, it requires estimating the shear velocity, which is not a trivial task in conceptual models and hence, needs iteration to determine the critical particle size given. Many othe empirical approaches have been used to estimate the incipient condition of sediment motion. The reader is referred to Beheshti & Ataie-Ashtiani (2008) regarding the empirical s in use. They share the same limitation as the of Soulsby for application in simple conceptual methods. Hjulström (1939) developed the famous(Southard, 2006) Hjulstrom diagram in the same period as the work of Shields. Hjulstrom diagram relates the critical flow velocity for incipient motion to the particle size. Due to the fact that it is dimensional and can be easily implemented in simple models, the incipient condition in our conceptual model depends on the Hjulstrom diagram. Miedema (2010) has fitted an empirical to the Hjulstrom diagram. The new conceptual model needs to determine the critical diameter of incipient motion corresponding to a given flow velocity thus the Miedema's empirical should be solved in the reverse direction and this requires iteration. To avoid this complication we fitted a separate empirical (Equation 2.2) to the resuspension curve of the Hjulstrom diagram. For similar reason as the resuspension, we fitted another empirical (Equation 2.3) to the deposition curve of Hjulstrom diagram. This enabled easy programming in our surrogate model and a straight forward evaluation of critical diameters, only based on the mean flow velocity, without having to directly quantify the critical shear stress. The mean flow velocity used in this research is simulated using the Muskingum

routing method of SWAT (Soil and Water Assessment

Tool) (Arnold et al., 1995).

where U is the mean flow velocity [m/s], $D_{*,f}$ and $D_{*,c}$

are the critical dimensionless grain diameters for the

resuspension of the fine and coarse part of the sedi

ment, respectively and $D_{*,depos1}$ and $D_{*,depos2}$ are the

critical dimensionless grain diameters for the deposi

tion corresponding to mean flow velocities less than

0.4 m/s and between 0.4 m/s and 1.94 m/s, respectively.

The two empirical s are applicable only for dimension

less grain sizes less than 1560. The particles sizes in

this study lie in this range.

The critical grain size of resuspension or depo

sition is determined as a function of the critical

dimensionless grain diameter [Equation (2.4)]

Where d_{cr} is the critical diameter of resuspension or

deposition; s[−] is the specific gravity of the sedi

ment, υ is the kinematic viscosity of water [m^2/s];

g is acceleration due to gravity [m/s^2] and $D*$ is the dimensionless grain size.

2.3 The particle size distribution

Sediment samples collected during a storm events as well as under dry flow conditions in rivers usu ally exhibit a lognormal distribution (Abuodha, 2003; Agrawal et al., 2012; Bouchez et al., 2011).

For each of the four categories of the sediment par ticle size distributions used as a boundary, we fitted a lognormal probability density function by calibrating the mean and standard deviation of the distribution. The log-normal distributions were transformed to nor mal distributions because the latter has more formulas available for computing the distribution parameters during the mixing of different samples and truncated distributions. The log-normal distribution of the parti cle sizes was represented using the probability density function shown in Equation (2.5).

where d is the particle diameter and it is always greater than zero[$*10^{-5}$ m]; μ is the mean of the log transformed particle diameters; and δ is the standard deviation of the log-transformed particle diameters. The mean diameter and standard deviation of the sediment particles is updated during each time step to account for mixing and deposition-resuspension processes. The parameters of two mixed normally distributed samples of suspended sediment samples were evaluated using Equation (2.6) & Equation (2.7). A mixture distribution is treated as a bimodal distribution only if the difference between the mean of the two normal distributions is greater than the sum of the two standard deviations (Schilling et al., 2002). We tested if the combinations of the particle size distributions from different boundaries satisfy the condition for unimodality. The masses of the two sediment samples in a mixture represent their respective weights when applying the s used for computing the mean size and standard deviation of the mixture distribution. where μ_{mix} is mean of the mixed samples; δ^2_{mix} is the variance of the mixed samples; μ_a & μ_b are variances of the two log-transformed sediment particle size distributions and p_a and p_b are the weights of the two samples. $p_a + p_b = 1$. For the sake of simplicity, the population mean and standard deviation were assumed to change according to the sample mean and standard deviation. After a deposition, the upper tail of the distribution extending up to the critical diameter of settlement was used to compute the mass of the settled sediments. In order to determine the statistical parameters of the mixture of normal distributions, it was important to quantify the mean and standard deviation of the truncated normal distribution using the statistical formulae published by Barr & Sherrill (1999) and Vernic et al. (2009). Accordingly, the mean and standard deviation of the upper truncated normal distribution were evaluated using Equation (2.8) & Equation (2.9), respectively. The mean and variance of the lower truncated normal distribution corresponding to the sediment settling to the bed were evaluated using Equation (2.10) & Equation (2.11), respectively. Where d_{cr} is the critical diameter[$*10^{-5}$ m] of deposition; $\beta = (\ln(d_{cr})-\mu)/\delta$; $\lambda(\beta) = \varphi(\beta)/[1 - \Phi(\beta)]$;

$\delta(\beta) = \lambda(\beta)/[\lambda(\beta) - \beta]$; $E(d|d < d_{cr})$ is the mean of log transformed particle sizes of the sediment remaining in suspension; $Var(d|d < d_{cr})$ is the variance of log transformed particle sizes of the sediment remaining in suspension; $E(d|d > d_{cr})$ is the mean of log transformed particle sizes of the settling sediment; $Var(d|d > d_{cr})$ is the variance of log transformed particle sizes of the settling sediment.

We calculated the fraction of sediment mass in the channel settling to the bottom using Equation (2.12).

Where Frac settl is the settling fraction of sediment mass in the channel; erfc is the complementary error function.

For a known settling fraction of sediment mass in the channel, we determined the corresponding critical diameter of the sediment using Equation (2.13).

Where erfinv is the inverse of error function.

Resuspension was assumed to take place only if the settled sediment is available in the bottom sediment reservoir. Therefore, no new detachment of sediments from the banks or bottom of the river was considered.

The mixing of distributions and updating of the moments of mixed distribution was also performed for the bottom reservoir. Similar to the sediment in the suspension, the particle sizes of the bottom sediment reservoir were also assumed to follow lognormal dis tribution. We determined the re-suspending fraction of the bottom sediment mass using Equation (2.14)

Where Frac resus is re-suspending fraction of the sedi ment mass from the bottom sediment reservoir; d cr,resus is the critical particle diameter for resuspension.

2.4 The sediment carrying capacity

The sediment transport capacity of a stream flow is the steady flux of sediments that the flow can transport (Prosser & Rustomji, 2000). In our model, the sedi ment transport capacity of a given flow is imposed by

using Velikanov's energy (Velikanov, 1954), as imple

mented by Zug et al. (1998) and used by Shrestha

(2013) [Equation (2.15)]. where CT min and CT max are the minimum and maximum sediment concentrations, respectively that the stream can carry; η 1 and η 2 are the critical sedimentation and erosion efficiency coefficients, respectively; s is the specific gravity of the sediment; ρw is the density of water, U is the mean flow velocity, ω s is the settling velocity and I is the channel slope. At the beginning of each calculation time step, the total mass of suspended sediment in each river reach is updated, by representing a reach as a continuously stirred reservoir. The latter mass is then compared to the sediment carrying capacity of the stream. If the sediment in suspension is less than the minimum carrying capacity, resuspension takes place, as governed by the re-entrainment criteria and the sediment mass available in the bottom reservoir. If the sediment in the suspension is less than the maximum carrying capacity but greater than the minimum carrying capacity, the critical condition of deposition is checked based on the critical diameter of settlement. If the sediment mass exceeds the maximum carrying capacity, the coarsest particle sizes are deposited. 2.5 The model comparison The new method, based on the particle size distribution and further called PSD method, was compared to the widely adopted method of sediment transport that assumes a complete mixing and calculates the suspended sediment mass using only the sediment carrying capacity (e.g. Neitsch et al., 2009; Viney & Sivapalan, 1999; Williams, 1980). The latter method is further called the SCC method. In the SCC model, a linear reservoir concept was used for complete mixing and transport of sediment and the Velikanov's energy was used for stream power (transport capacity). Unlike the PSD method, the SCC method does not impose the critical diameter condition for deposition and resuspension. For a fair comparison, resuspension in both models was enabled only when there was sufficient deposited sediment mass in the bottom reservoir. 3 RESULTS AND DISCUSSION Based on experimental data for the Zenne basin during dry weather periods (Shrestha, 2013), log-normal distribution functions have been fitted to the different boundaries of the river system (Table 3.1). Table 1. Mean and standard deviations of the logtransformed normal distributions of particle sizes(*10 −5 m). Boundary Location parameter Shape parameter CSOs 1.75 1.08 WWTPs

2.26 0.69 Tributaries 1.00 1.80 Canal 0.20 1.10

Figure 1. The new empirical s reconstruct the Hjulstrom diagram(digitized from (Miedema, 2010)) with good agreement.

Figure 2. The PSD model simulates the low sediment concentrations better than the conventional conceptual method(SCC)

both at Vilvoorde and Eppegem stations. The overestimation of the high concentrations by the PSD model is considered as an

affirmative trend given the distribution of the better quality observed data for the period not covered in the simulation.

As long as the difference between the mean val

ues of two normal distributions is not greater than the

sum of the respective standard deviations, the mixture

distribution would be considered unimodal (Schilling

et al., 2002). In this regard, only the difference between

the mean of the log-transformed normal distribution of

WWTPs (waste water treatment plants) and the canal

is slightly greater than the sum of the respective stan

dard deviations (Table 3.1). Therefore, a mixture of

sediment between any two of the four boundaries was

assumed unimodal.

The two empirical equations we fitted to the Hjul

strom diagram reproduces the deposition and resus

pension curves(Fig. 1).

A comparison of the simulated sediment concen

trations by the new method and the SCC approach, after comparable calibration efforts, revealed that

both methods simulate comparable low concentrations (Fig. 2). Both methods seem to slightly overestimate the low concentrations compared to the VMM observations. The PSD method simulated much higher high sediment concentrations than the SCC method. Before making any judgement of which method performed better, it is important to assess the reliability of the observed data. The observed peak concentrations from the VMM are significantly lower as compared to observations with more recent data of much higher temporal resolution that were obtained from the Flemish Hydraulics Research institute (FHR).The latter institute takes sediment samples every seven hours from representative depths, while the VMM samples were collected with buckets from the water surface. Studies show that sediment concentration increases with depth (Agrawal et al., 2012; Bouchez et al., 2011; van Rijn, 1993)

Figure 3. The box plot of the sediment concentrations sim

ulated by the PSD method agrees with the distribution of the

good quality sediment concentration of FHR but the SCC

method seriously underestimates the upper quantiles.

and hence concentration near the water surface are

normally significantly lower than the depth averaged

concentration. Based on this fact, we believe that the

data from FHR are more realistic and that the actual

peak concentrations are thus probably higher than what

the VMM data suggest.

Unfortunately, the FHR data do not overlap with the

simulation period used in this research. Thus we could

not make quantitative evaluation of the model perfor

mance against the new data set; instead we used the

analogy from comparison of the overlapping dataset of

both data sources for the recent period (2011–2014)

(Fig. 3). From the comparison, the VMM observed peak sediment concentrations turned out to be signif icantly underestimated. Given the fact that the period of simulation (2007–2010) and the good quality FHR data (2011–2014) are close to each other and there is no significant difference in the rainfall and discharge of the two periods, it is normal to expect similar distri butions of sediment concentrations for both periods. From the analogy, the VMM data used in this research for the earlier period (2007–2010) were also underes timated (Fig. 3). Based on this fact, we considered the overestimation of the simulated peak sediment concen trations by the new simulator compared to the VMM data as an affirmative trend. The quantile compari son of the box plot at Eppegem station shows that

there is a good agreement for high concentrations and range of distribution between the PSD simulated concentrations and the FHR data (Fig. 3). The SCC method on the other hand simulated seriously underestimated high concentrations compared to the FHR data (Figs. 2 & 3). Both models show slight overestimation of the low concentrations compared to the VMM data (Fig. 2). Given the fact that the low concentrations of the VMM data are slightly underestimated compared to the FHR data (Fig. 3), the simulations of the low concentrations by both models is acceptable. The SCC method simulates very narrow range of concentration compared to the concentration distribution of the good quality data of FHR (Fig. 3). The advantage of the PSD model for simulating realistic high concentrations and similar concentration range as the good quality data of FHR makes it advantageous over the SCC method. The advantage of the PSD method over the SCC method is attributed to the fact that the PSD method updates the median size of the sediments at every time step and thus calculates a realistic settling velocity and hence the sediment carrying capacity of the river. Besides imposing the carrying capacity as a limit, it imposes the critical condition for the incipient motion, and this contributed for a better performance. 4 CONCLUSIONS Quantifying the fine proportion of the suspended sediment load is essential for investigating the transport of adsorbed pollutants. The usual approaches in hydrodynamic models are computationally expensive and the approaches in conceptual sediment transport models do not account for the distributions of sediment particle sizes and their dynamics (Viney and Sivapalan, 1999; Willems, 2010). We presented an analytic solution using log-normal probability density functions to represent the particle size distributions of sediment. The empirical frequency distributions of the sediment particle sizes of the Zenne River was acceptably represented by log-normal distribution functions. The proposed conceptual model applies the critical condition for the initiation of motion and deposition by means of a simple algebraic representing the Hjustrom diagram. The eroded or deposited sediment mass is determined based on an analytical evaluation of the area under the probability density function extending from the tail to the critical sediment size of deposition/resuspension. A comparison of the simulation result from the new method and the conventional CSTR – sediment carrying capacity (SCC) based approach at two measuring stations along the Zenne River shows that the new method performs better than the SCC method when qualitatively compared to the distribution of high sediment concentrations and the range of the observation of the good quality data from FHR. The SCC method indeed systematically underestimated the high concentrations and simulated very narrow concentration

range while performing equally good with the PSD

method for the low concentrations.

The PSD method simulated the extreme high con

centrations better than the conventional SCC method

because it accounts for a dynamic median size of the

sediments and thus adapts realistic settling velocity

and hence the sediment carrying capacity of the river.

It also imposes the critical condition for the initiation of motion, apart from the carring capacity and this contributed for better performance.

Inspite of the promising performance of the new simulator, the particle size distributions used in this study are limited to samples collected during low and average flow conditions and thus do not represent the storm conditions due to data limitation. Accounting for the seasonal variability of the particle size dis tributions could be a subject of future researches. In cases where the particle size does not follow a uni modal distribution, the new PSD method might not give realistic simulation results because it is based on the assumption of normal distribution.

We believe that this work enhances the simulation of sediment-adsorbed pollutant transport in simple conceptual water quality models.

ACKNOWLEDGEMENT

The authors acknowledge that the presented work was developed in the framework of the project “Develop ment of conceptual models for an integrated river basin management” financed and coordinated by Flanders Hydraulics Research.

Abuodha, J.O.Z., 2003. Grain size distribution and com position of modern dune and beach sediments, Malindi Bay coast, Kenya. J. African Earth Sci. 36, 41–54.

doi:10.1016/S0899-5362(03)00016-2

Agrawal,Y.C., Mikkelsen, O. a, Pottsmith, H.C., 2012. Grain

Size Distribution and Sediment Flux Structure in a River

Profile , Measured with a LISST-SL Instrument. Bellevue,

USA.

Ani, E.C., Hutchins, M., Kraslawski, A., Agachi, P.S.,

2011. Mathematical model to identify nitrogen variabil

ity in large rivers. River Res. Appl. 27, 1216–1236.

doi:10.1002/rra.1418

Arnold, J., Williams, J., Maidment, D., 1995. Continuous

Time Water and Sediment-Routing Model for Large

Basins. J. Hydraul. Eng. 121, 171–183. doi:10.1061/

(ASCE)0733-9429(1995)121:2(171)

Asselman, N.E.M., 2000. Fitting and interpretation of

sediment rating curves. J. Hydrol. 234, 228–248.

doi:10.1016/S0022-1694(00)00253-5

Barr, D.R., Sherrill, E.T., 1999. Mean and Variance of

Truncated Normal Distributions. Am. Stat. 53, 357–361.

doi:10.1080/00031305.1999.10474490

Beheshti, a. a.,Ataie-Ashtiani, B., 2008.Analysis of
threshold

and incipient conditions for sediment movement. Coast.

Eng. 55, 423–430. doi:10.1016/j.coastaleng.2008.01.003
Bertrand-Krajewski, J.-L., 2006. Modelling of sewer solids
production and transport. Cours DEA “Hydrologie Urbaine.
Bouchez, J., Metivier, F., Lupker, M., Maurice, L., Perez,
M., Gaillardet, J., France-Lanord, C., 2011. Prediction of
depth-integrated fluxes of suspended sediment in the Amazon
River: particle aggregation as a complicating factor.

Hydrol. Process. 25, 778–794. doi:10.1002/hyp.7868 Buscombe, D., Rubin, D.M., Lacy, J.R., Storlazzi, C.D., Hatcher, G., Chezar, H., Wyland, R., Sherwood, C.R., 2014. Autonomous bed-sediment imaging-systems for revealing temporal variability of grain size. Limnol. Oceanogr. Methods 12, 390–406. doi:10.4319/lom.2014. 12.390 Church, M., Krishnappan, B.G., 1998. Sediment sources, transport processes, and modeling approaches for the Fraser River in C. Heal. Fraser River Aquat. Ecosyst. Environ. Canada, DOE FRAP 11, 63–80. Corbett, D.R., 2010. Resuspension and estuarine nutrient cycling: Insights from the Neuse River Estuary. Biogeosciences 7, 3289–3300. doi:10.5194/bg-7-3289-2010 Crabill, C., Donald, R., Snelling, J., Foust, R., Southam, G., 1999. The impact of sediment fecal coliform reservoirs on seasonal water quality in Oak Creek, ARIZONA. Water Res. 33, 2163–2171. doi:10.1016/S0043-1354(98) 00437-0 Dhi, 2007. Mike 11 reference manual. Guo, S., Zhang, F., Wang, B., Zhang, C., 2012. Settlement prediction model of slurry suspension based on sedimentation rate attenuation. Water Sci. Eng. 5, 79–92. doi:10.3882/j.issn.1674-2370.2012.01.008 Hjulström, F., 1939. Transportation of detritus by moving water: Part 1. Transportation. AAPG Spec. Vol. Jain, R.K., Kothyari, U.C., 2010. Influence of cohesion on suspended load transport of non-uniform sediments. J. Hydraul. Res. 48, 33–43. doi:10.1080/ 00221681003696317 Jamieson, R., Joy, D., D., M., Lee, H., Kostaschuk, R., Gordon, R., 2005. Transport and deposition of sedimentassociated Escherichia coli in natural streams. Water Res. 39, 2665–2675. Kumar, S., Rastogi, R.A., 1987. A conceptual catchment model for estimating suspended sediment flow. J. Hydrol. 95, 155–163. doi:10.1016/0022-1694(87)90122-3 Lewis, D.J., Atwill, E.R., Pereira, M.D.G.C., Bond, R., 2013. Spatial and Temporal Dynamics of Fecal Coliform and Associated with Suspended Solids and Water within Five Northern California Estuaries. J. Environ. Qual. 42, 229. doi:10.2134/jeq2011.0479 Miedema, S.A., 2010. Constructing the Shields curve, a new theoretical approach and its applications, in: WODCON XIX. WODCON XIX, Beijing China. Miller, G.A., Amemiya, M., Jolly, R.W., Melvin, S.W., Nowak, P.J., 1982. Soil Erosion and the Iowa Soil 2000 Program (Staff General Research Paper No. 11462). Iowa State University, Department of Economics. Neitsch, S.L., Arnold, J.G., Kiniry, J.R., Williams, J.R., 2009. Soil & Water Assessment Tool – Theoretical Documentation Version 2009. Texas Water Resources Institute, Texas. Ongley, O.E.D., 1996. Control of water pollution from agriculture FAO irrigation and drainage paper 55. Food and Agriculture Organization of the United Nations, Rome. Ouattara, N., Passerat, J., Servais, P., 2011. Faecal contamination of water and sediment in the rivers of the Scheldt drainage network. Environ. Monit. Assess. 1-15. Paphitis, D., 2001. Sediment movement under unidirectional flows: An assessment of empirical threshold curves. Coast. Eng. 43, 227-245. doi:10.1016/S0378 3839(01)00015-1

Prosser, I.P., Rustomji, P., 2000. Sediment transport capac ity relations for overland flow. Prog. Phys. Geogr. 24, 179-193. doi:10.1177/030913330002400202

Sartor, J., 1972. Water Pollution Aspects of Street Surface Contaminants. U.S. Environmental protection agency, Washington, D.C.

Schilling, M.F., Watkins, A.E., Watkins, W., 2002. Is Human Height Bimodal? Am. Stat. 56, 223-229. doi:10.1198/00031300265

Shields, A., 1936. Application of Similarity Principles and Turbulence Research to Bed-Load Movement, Mitt. Preuss. Versuchsanst. Wasserbau Schiffbau.

Shrestha, N.K., 2013. Development of OpenMI compliant simulators for integrated water quality management, with application on the River Zenne (Belgium), PhD thesis. Vrije Universiteit Brussel, Brussels.

Soulsby, R.L., Whitehouse, R.J.., 1997. Threshold of sedi ment motion in coastal environments. Proc. Pacific Coasts Ports 1997 Conf. 1, 149-154.

Southard, J., 2006. Threshold of Movement. Spec. Top. An Introd. to Fluid Motions, Sediment Transp. Curr.

Sediment. Struct. 260–284.

Van Rijn, L.C., 1993. Simple general formulae for sand

transport in rivers, estuaries and coastal waters [WWW

Document]. URL http://www.leovanrijn-sediment.com/

Figure. The theoretical probability distribution fitted to the empirical cumulative distribution of the particle sizes

Velikanov, M.A., 1954. Gravitational Theory of Sediment Transport. J. Sci. Sov. Union, Geophys. Russ. 4. Vernic, R., Teodorescu, S., Pelican, E., 2009. Two Lognormal Models for Real Data. Viney, N.R., Sivapalan, M., 1999. A conceptual model of sediment transport: application to the Avon River Basin in Western Australia Abstract: Hydrol. Process. 743, 727–743. Viney, N.R., Sivapalan, M., Deeley, D., 2000. A conceptual model of nutrient mobilisation and transport applicable at large catchment scales. J. Hydrol. 240, 23–44. doi:10.1016/S0022-1694(00)00320-6 Willems, P., 2010. Parsimonious Model for Combined Sewer Overflow Pollution. J. Environ. Eng. 136, 316–325. doi:10.1061/(ASCE)EE.1943-7870.0000151 Williams, J.R., 1980. Spnm, a Model for Predicting Sediment, Phosphorus, and Nitrogen Yields from Agricultural Basins1. JAWRA J. Am. Water Resour. Assoc. 16, 843–848. doi:10.1111/j.1752-1688.1980.tb02497.x Young, R. a., Onstad, C. a., Bosch, D.D.,Anderson,W.P., 1989. AGNPS: A Non-Point Source Pollution Model for Evaluating Agricultural Watersheds. J. Soil Water Conserv. 44, 168–173. Zug, M., Bellefleur, D., Phan, L., Scrivener, O., 1998. Sediment transport model in sewer networks – a new utilisation of the Velikanov model. Water Sci. Technol. 37, 187–196. doi:10.1016/S0273-1223(97)00769-5 Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Effect of seepage on the friction factor in an alluvial channel

P. Mahesh, V. Deshpande, & B. Kumar

Department of Civil Engineering,

Indian Institute of Technology Guwahati, India

ABSTRACT: In this study, an experimental approach is used to observe the effects of downward seepage on flow resistance in a threshold alluvial channel with parabolic cross sectional shape. Experiments are performed over fine and coarse sand bed channels in two categories: no seepage experiment and seepage experiment. Shear velocities are evaluated from the Reynolds shear stress distribution for both sets of experiments. Increase in the value of shear velocity is observed for the seepage experiment. Log-law of the velocity distribution in alluvial channel is modified over rough boundary with parabolic cross-sectional shape of the alluvial channel.The value of von-Karman’s constant is decreased with the application of downward seepage. Increase in turbulent production and decrease in turbulent kinetic energy dissipation are observed in the presence of downward seepage. Further, Darcy-Weisbach friction factor is calculated and increase in the flow resistance is observed under the action of downward seepage.

1 INTRODUCTION

Alluvial channels often encounter permeable bound aries in natural environments such as porous bound aries consisting of sediment particles in natural rivers and in irrigation canals. Exchange of water (seepage) through the porous boundaries can occur in either way, flow from the channel (downward seepage) or into the channel, depending upon the difference of level between the water in the channel and the surrounding groundwater table. Earthen irrigation canals lose con siderable amount of water through the bed and sides of the canals resulting in low conveyance efficiency

(Yussuff et al. 1994, Berenbrock 1999, Carlson and Petrich 1999, Tanji and Kielen 2002, Fipps 2005, Kinzli et al. 2010, Martin and Gates 2014).

Resistance to flow offered by the granular bound aries, plays a significant role in an alluvial channel. Therefore, the adequate knowledge of friction factor is required for the design of an alluvial channel. Var ious experimental and field studies have been carried out for evaluating the resistance to flow in the allu vial channel (Einstein and Barbarossa 1952, Te Chow 1959, Rouse 1965, Engelund 1966). Rouse (1965) has divided the flow resistance in different forms such as: grain resistance, form resistance, wave resistance, and resistance caused by the flow unsteadiness. Song et al. (1998) observed an increase in the friction factor for mobile bed flows. Some empirical studies (Yen 2002, Rao and Kumar 2009) were performed to predict the friction factor over the rough boundary. Dey et al. (2012) evaluated the Darcy-Weisbach friction factor by using turbulent flow structure for clear water and mobile bed flows. They observed that the resistance to flow in clear water was higher as compared with mobile bed flow. Recently, some studies (Patel et al. 2015, Deshpande and Kumar 2016) have investigated that timemean velocities and Reynolds shear stresses increased because of downward seepage in coarse grained sandbed channel. In addition to this, Deshpande & Kumar (2016) observed that the value of von-Karman's constant was decreased when water was extracted in the downward direction from the channel bed. However, understanding of the interaction between groundwater and surface water is necessary in order to find its influence on the friction factor. Thus, the objective of this study is to investigate the friction factor over sand bed channel in the presence of downward seepage. Further, present work also focuses on the variation in the value of von-Karman's constant in alluvial channel under the action of downward seepage. 2 EXPERIMENTAL SET-UP AND METHODOLOGY Experiments were performed in a 20 m long, 1 m wide, and 0.72 m deep recirculating transparent plexiglassed tilting flume. Schematic diagram of the tilting flume is shown in Figure 1. More details of experimental setup and facility can be found in the work of Patel et al. (2015). Experiments were performed on two uniform river sands: fine grained (d_{50} = 0.41 mm) and coarse grained (d_{50} = 1.1 mm). The geometric standard deviation (Marsh et al. 2004) and angle of repose were obtained as 1.17 (1.03), and 32.55 $^\circ$ (31.15 $^\circ$), respectively, for the fine (coarse) grained sands used in this study. A parabolic cross-sectional profile of the channel was made based on Lane's (1953) theory,

Figure 1. Schematic diagram of the tilting flume.

Figure 2. Vertical distributions of Reynolds shear stresses against normalized depth ($h^+ = z/h$) (a) for fine sand bed channel

(Run 1) and (b) for coarse sand bed channel (Run 2). Where z is distance from bed surface and h is the center line flow depth.

importance of which is to design a stable channel at

the incipient motion condition of boundary particles.

In the present experiments, the cross-sectional shape of

the main channel was evaluated for fine (coarse) sand

by using the maximum top width and center line depth

as 0.70 m (0.70 m) and 0.14 m (0.135 m), respectively. Experimental framework was designed for two scenarios: no seepage experiment and seepage experiment. In the no seepage experimental run, main channel discharge was kept in such a way that the bed particles were on the threshold of motion over the entire test length of the channel.

Incipient motion

condition was obtained by using Yalin's (1976) crite rion during the no seepage experiment. In accordance with the literature (Berenbrock 1999, Carlson and Pet rich 1999, Martin and Gates 2014), 15% and 30% of main channel discharge were chosen for fine and coarse grained sand bed channel as downward seep age, respectively. Channel bed slope for experiments on both sands was 0.00176. Main channel discharge at the incipient motion condition during the no seep age experimental run was 0.0169 m^3/s (0.0209 m^3/s) for Run 1 (Run 2). Where 'Run 1' and 'Run 2' indi cate experiments over fine and coarse grained bed, respectively.

Instantaneous flow measurements were taken with the help of Vectrino + Acoustic Doppler Velocimeter developed by Nortek. Samples were collected for five minutes duration at a sampling rate of 100 Hz. Around 20–25 velocity samples were recorded in the verti cal profile at the middle of the test section (8 m from downstream reach end) for both no seepage experi ment and seepage experiment (immediately after the application of downward seepage). Signal to noise ratio (SNR > 15) and correlation (>70) were used for filtering the raw velocity samples. A spike removal technique (Goring and Nikora 2002) was used such that the Kolmogorove's −5/3 scaling law was satisfied in the inertial subrange with acceleration threshold values ranging from 1 to 1.5.

3 RESULTS

At any point in the flow domain, time-mean veloci ties and Reynolds stresses can be obtained from the decomposition of the instantaneous velocity samples (u_i) into the time-mean (u) and fluctuating components of velocity (u') (Tennekes and Lumley 1972). Reyn ods shear stresses (RSS) provide information about the momentum transfer in the vertical direction of the flow. Figure 2 shows the vertical distribution of RSS for no seepage and seepage scenarios, plotted against the normalized depth of flow ($h^+ = z/h$), where z is distance from bed surface (positive in the vertically upward direction) and h is the center line flow depth. Shear velocities (u_{*t}) were calculated by extending the linear portion of Reynolds shear stresses on the chan nel bed for no seepage and seepage experiments by using the following expression (Nezu and Nakagawa 1993):

From Figure 2, we can observe that Reynolds shear stresses are increased for seepage experiment (imme diately after the application of seepage), suggesting an increased momentum transfer toward the channel bed under the action of downward seepage. In turbulent flows, shear velocity defines the scale for fluctuating component of the velocity. The values of shear veloc ities are evaluated as 0.016 (0.023 m/s) m/s and 0.019 (0.026) m/s over fine (coarse) sand bed for no seepage

Figure 3. Distributions of the non-dimensional time-mean velocities ($u^+ = u/u_{*t}$) for the no seepage and seepage experiments. Equations are given for the modified log-law over sand bed channel for the no seepage and seepage scenarios. and seepage experiment, respectively. A higher value of shear velocity for seepage experiment indicates that scales of velocity fluctuations in the flow are increased in the presence of downward seepage. Figure 3 shows the vertical distributions of the nondimensional time-mean streamwise velocities for no seepage and seepage experiments on both sands. The law of wall is applicable only for lower 20% of velocity data points (Nezu and Nakagawa 1993). Therefore, a logarithmic trendline is fitted to the inner layer ($h^+ < 0.2$) of each velocity profile as has been suggested by Dey et al. (2012). Time-mean velocity (u) and the distance of the velocity measurement point from the bed surface (z) are normalized as $u^+ = u/u_{*t}$ and $z^+ = z/d_{50}$, respectively. The data has been plotted by using the following non-dimensional form of the log-law: where ▯z is the depth of virtual bed level (▯z^+ =▯z/d_{50}), z_0 is zero velocity level and k is von-Karman’s constant. The values of ▯z and z_0 are determined by the least square method, which correspond to the maximum value of the correlation coefficient. We can observe that the universal logarithmic law of wall for flows over rough boundaries does not apply for the flow over parabolic cross-sectional sand bed channel. Equation of the universal logarithmic law ($u^+ = 2.43 * \ln(z^+) + 8.5$) has been modified for no seepage experiment, considering the flow in a channel with curved cross-section. Because of the strong exchange of momentum between the flow over the channel bed and the seepage flow, the modified logarithmic law for no seepage experiment had to be further modified considering the downward seepage scenario. Average values of ▯z, k , and z_0 for no seepage and seepage experiments are given in Table 1. We can observe from Table 1 that for the no seepage experiment, value of von-Karman’s constant agrees

well with the universal value. However, for the seep age experiment the value of von-Karman's constant is reduced. Reduction in the value of k is in agreement with the increased sediment transport, which occurred in the channel after the application of downward seep age. In addition, values of the zero velocity level and depth of the virtual bed level are increased when water was extracted from the channel in the downward direc tion, which exhibits the exposure of bed particles to a higher velocity zone.

Figure 4 depicts the vertical distributions of turbu lent production and turbulent kinetic energy dissipa tion for experiments on both sands under no seepage and seepage conditions, which can be evaluated as (Krogstadt and Antonia 1999)

Table 1. Average values of Δz, k , and z 0 .

Scenario	Δz	k	z 0
No seepage	5.1 mm	0.40	0.07 mm
Seepage	6.78 mm	0.33	0.27 mm

Figure 4. Vertical distributions of turbulent production (T P) and turbulent kinetic energy dissipation (E D) for no seepage and

seepage experiments over (a) fine sand bed channel (Run 1) and (b) coarse sand bed channel (Run 2). where υ is the kinematic viscosity. Quantities t p and e d are non-dimensionalized by multiplying with h/u 3 *t , and are expressed as T P and E D , respectively. Turbulent production comes from the interchange of mean flow energy to fluctuations. We can observe from Figure 4 that the turbulent production is increased when the downward seepage is applied to the channel. This increased T P is in agreement with the higher momentum transfer under the seepage condition. Also, it can be implied that larger roughness over the boundary of the channel with the application of seepage, causes more turbulent production in the flow. Further, we can observe that the turbulent dissipation (E D) is reduced when water is extracted in the downward direction from the sand bed. Under the action of downward seepage, more energy is converted to turbulent fluctuations in the flow, resulting in the reduced dissipation of the turbulent kinetic energy. In order to determine the friction factor in flow, Darcy-Weisbach relation is used as: where λ is the friction factor and U is the mean velocity. The mean velocity is obtained by integrating the velocity profile over the flow depth for no seepage and seepage experimental scenarios. Additionally, Raudkivi (1967) and Rao & Kumar (2009) have

suggested that the velocity corresponding to z = h/e can be considered as the mean velocity of the flow in an open channel. The values for friction factor are calculated as 0.024 (0.039) and 0.031 (0.047) for no seepage and seepage experiment over fine (coarse) sand bed channel, respectively. This increase in the value of friction factor after the application of down ward seepage indicates that downward seepage causes higher roughness on the channel bed and the resistance to flow is significantly increased under the condition of downward seepage.

4 CONCLUSIONS

In the present study, an experimental approach was used to observe the effects of downward seepage on flow resistance in a threshold alluvial channel with parabolic cross-section. Experiments were performed in two categories: (1) when there was no seepage present and the channel was on the threshold of sed iment movement (no seepage experiment), and (2) when water was extracted in the downward direction from the threshold channel through the channel bound ary (seepage experiment), without any increment in inflow discharge. In the seepage experiment, Reynolds shear stresses and shear velocities were increased, which indicate increase in the momentum transfer toward the channel bed. We observed that the value of von-Karman's constant decreased, which is associated with increased bed material transport under the con dition of downward seepage. Modified law of wall for the flow in parabolic cross-sectional sand bed channel is introduced when water is extracted in the vertically downward direction from the channel bed. Increase in turbulent production and reduced turbulent kinetic energy dissipation in the presence of seepage have been observed. Analysis of Darcy-Weisbach friction factor shows that it increases significantly with the application of seepage, causing more resistance to the flow.

ACKNOWLEDGMENT

The authors gratefully acknowledge the financial sup port that was received from the Department of Sci ence and Technology, Govt. of India (SERC-DST:

SR/S3/MERC/005/2010) to carry out the research

work presented in this paper.

Berenbrock, C. (1999). Streamflow gains and losses in the

lower boise river basin, idaho, 1996–97. Technical Report

99–4105.

Carlson, R. & C. Petrich (1999). New york canal geologic

cross-section, seepage gain/loss data, and ground water

hydrographs: compilation and interim findings. Technical

Report IWRRI-2002-09. Deshpande, V. & B. Kumar (2016). Turbulent flow structures in alluvial channels with curved cross-sections under conditions of downward seepage. Earth Surface Processes and Landforms. Dey, S., R. Das, R. Gaudio, & S. K. Bose (2012). Turbulence in mobile-bed streams. Acta Geophysica 60(6), 1547–1588. Einstein, H. A. & N. L. Barbarossa (1952). River channel roughness. Transactions of the American Society of civil Engineers 117(1), 1121–1132. Engelund, F. (1966). Hydraulic resistance of alluvial streams. Journal of the Hydraulics Division 92(2), 315–326. Fipps, G. (2005). Potential water savings in irrigated agriculture for the rio grande planning region (region m). Texas Cooperative Extension, Texas A&M University System. Goring, D. G. & V. I. Nikora (2002). Despiking acoustic doppler velocimeter data. Journal of Hydraulic Engineering 128(1), 117–126. Kinzli, K.-D., M. Martinez, R. Oad, A. Prior, & D. Gensler (2010). Using an adcp to determine canal seepage loss in an irrigation district. Agricultural water management 97(6), 801–810. Krogstadt, P.-Å. & R. Antonia (1999). Surface roughness effects in turbulent boundary layers. Experiments in Fluids 27(5), 450–460. Lane, E. W. (1953). Progress report on studies on the design of stable channels by the bureau of reclamation. Volume 79, Separate 280, pp. 1–31. American Society of Civil Engineers. Marsh, N. A., A. W. Western, & R. B. Grayson (2004). Comparison of methods for predicting incipient motion for sand beds. Journal of Hydraulic Engineering 130(7), 616–621. Martin, C. A. & T. K. Gates (2014). Uncertainty of canal seepage losses estimated using flowing water balance with acoustic doppler devices. Journal of Hydrology 517, 746– 761. Nezu, I. & H. Nakagawa (1993).Turbulence in Open Channel Flows. Balkema,

Rotterdam, The Netherlands. Patel, M., V. Deshpande, & B. Kumar (2015). Turbulent characteristics and evolution of sheet flow in an alluvial channel with downward seepage. Geomorphology 248, 161–171. Rao, R. & B. Kumar (2009).Analytical formulation of the correction factor applied in einstein and barbarossa equation (1952). Journal of Hydrology and Hydromechanics 57(1), 40–44. Raudkivi, A. J. (1967). Loose Boundary Hydraulics. Pergamon press. Rouse, H. (1965). Critical analysis of open-channel resistance. Journal of the Hydraulics Division 91(4), 1–23. Song, T., Y. Chiew, & C. Chin (1998). Effect of bed-load movement on flow friction factor. Journal of Hydraulic Engineering 124(2), 165–175. Tanji, K. K. & N. C. Kielen (2002). Agricultural drainage water management in arid and semi-arid areas. FAO irrigation and drainage paper. Te Chow, V. (1959). Open channel hydraulics. Tennekes, H. & J. L. Lumley (1972). A first course in turbulence. MIT press. Yalin, M. S. (1976). Mechanics of sediment transport. Pergamon press. Yen, B. C. (2002). Open channel flow resistance. Journal of hydraulic engineering 128(1), 20–39. Yussuff, S., H. Chauhan, M. Kumar, & V. Srivastava (1994). Transient canal seepage to sloping aquifer. Journal of Irrigation and Drainage Engineering 120(1), 97–109.
Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Experimental investigation of light particles transport in a tidal bore generated in a flume

L. Thomas, W. Reichl, Y. Devaux, A. Beaudouin & L. David

Axe HydEE, Département D2, Institut Pprime

Université de Poitiers, CNRS, ISAE-ENSMA, Poitiers, France

ABSTRACT: Tidal bores are natural flows that can have a strong impact on river ecosystems. Recently, there has been a regain of interest in this flow, for example in China. Many in-situ measurement campaigns have been performed, but they are very limited concerning the spatial resolution. Studies in the lab inside flumes, although questionable concerning their similitude to real tidal bores, can give new insights to understand the impact of

such phenomena on the river sediments. In this study, optical measurement techniques (stereo-PIV and PTV coupled) are used to add some understanding of the influence of moving bores on the sediment transport. Four Froude numbers are investigated. The trajectories and their stability are observed, and it is shown that for undular bores, the impact on the trajectories is weak, while strong effects are observed in the breaking bore case.

1 INTRODUCTION

Tidal bores are natural flows that can have a strong impact on rivers ecosystems. Although a great deal of work has been devoted to these interactions, this complex phenomenon still retains an unexplained part for which new insights and new tools may be of some interest. Many investigations have studied the hydro dynamic interactions between the river flow and the tidal bore and their impacts on the sediment trans port (Chen (2003), Chanson (2005), Simon et al. (2011), Fan et al. (2012)). The passage of the tidal bore generates a mixing zone of fine sediments (Kjerfve and Ferreira (1993), Wolanski et al. (2004), Chanson (2005)). These studies highlight scouring the muddy bottom and suspending materials for the passage of the bore. The undular bores induce the most intense sediment transport, as the whelps help keep sediment suspended Chanson and Tan (2010). The authors high light for each type of bore, the possible movements of the particles. For an undular bore, particles have a heli cal path with maximum vertical rise in the ridges and advection downstream in the hollows. For the break ing bore, in the upper part of the flow, the particles have a pseudo-chaotic movement and near the bottom, the particles have the same behavior as to a undular bore. Recently in laboratory, David et al. (014a) and David et al. (014b) have measured the instantaneous velocity fields during the arrival of bore and showed the different turbulent flows inside the water column in regards to several flow generation conditions.To better understand the impact of such phenomena on the sed iment transport, laboratory measurements are carried out with two simultaneous optical technics (stereo-PIV and PTV coupled). In the paper, the experiment will be first presented and will follow by description of the measurement technics. The hydrodynamic results will be after presented in an Eulerian and a Lagrangian point of view for four different flow cases. Finally the sediment transport will be characterized and compared to the hydrodynamic characteristics. 2 EXPERIMENT 2.1 Experimental set-up For the experiment we used an open water channel with a length of 8 m and a width of 0.4 m which can be seen in figure 1. The used worm pump had a speed control so that the flow rate could be fixed. The homogeneous behaviour of the inlet flow was ensured by a honeycomb grid followed by a convergent. In order to create a wave similar to the tidal bore in nature, a shutter door had been installed downstream of the measurement.This door was automatically moving downwards into the flow. The resulting increase of water depth led to the creation of a wave travelling upstream. This closing door was also used as a trigger for the other measurements. The first of these measurements gave the water level at different positions over time. For this purpose, acoustic sensors had been installed on top of the channel, two upstream of the PIV field and two downstream. These have been recording at a rate of 200 Hz. To get the necessary data for PIV and PTV, a laser sheet was used which was located at the centre of the channel parallel to the side walls. The used Darwin Duo Laser created a double pulse at 500 Hz with 15 mJ energy per single pulse. For the TR-PIV (Time Resolved PIV), two cameras of the type Photron SA1

Figure 1. Sketch of the experimental setup.

Table 1. Particles used for the measurements and filter used to separate the fluorescent signals.

Material	density	diameter	dye	filter
Vestosint	1.06	20 µm	Rhodamine6G	550 µm
PMMA	1.2	50 µm	Nile Red	610 µm
PMMA	1.2	300 µm	Nile Red	610 µm

with ten bits have been used, located at both sides of the channel, with an angle of about 20 ∘ . These two have been set up according to the Scheimpflug con dition. For the PTV measurement, a camera of type Photron 8 bits APX RS has been used which has been positioned perpendicular to the flow.

2.2 Particles

For the measurements, three types of particles have been used with two types of fluorescent colour applied to them. Moreover, two band filters have been found to match the wavelengths of these colours. All these are shown in table 1. The coating of these particles as well as the testing with different filters have been done in the lab by Patrick Braud. Physically speaking, the laser beams are hitting the coated particles which are diffusing the light over a certain range of wave length. The band filters are connected to the lenses of the cameras so that they only record part of the sig nal. Figure 2 shows the diffused light of the PMMA particles marked with Rhodamine B as well as the two used filters. Figure 3 shows the diffused light of the

Figure 2. Emission spectrum of the Rhodamine 6G, and signal received by the cameras through the different optical filters.

PMMA particles marked with Nile-Red and again the two filters. The 550nm filter will be used for particles marked with Rhodamine B and the 610nm filter will be used for particles marked with Nile-Red. In both figures it can be seen that the particles can also be seen on the wrong filter even though with a much lower intensity. This error needed to be corrected by some later filtering during the post-processing, using thresholding.

2.3 Flow configurations

A tidal bore is a positive surge travelling upstream, it is also called a hydraulic jump in translation. If one moves with the bore, ie the hydraulic jump is fixed in a translating reference frame, the surge can be steadily followed. The upstream velocity and the bore celerity

Figure 3. Emission spectrum of the Rhodamine 6G, and signal received by the cameras through the different optical filters.

Table 2. Particles used for the measurements and filter used to separate the fluorescent signals.

h (m)	U_f (m/s)	U_b (m/s)	Fr	type
0.118	0.353	0.859	1.13	undular
0.0940	0.665	0.595	1.31	mixed
0.0735	0.567	0.644	1.44	mixed
0.1435	0.871	1.01	1.58	breaking

are respectively U f and U b and the upstream water depth is h. The Froude number based on the initial water elevation is calculated in the moving frame as follows:

Four flow configurations corresponding two four dif ferent Froude numbers have been investigated. Three of them are similar to previous studies (David et al. (014a)) and the last is for a low Froude number with undular characteristics. They are summarized in table 2.

2.4 Image processing

• stereo-PIV The images were first cleaned using a background

subtraction, a Laplacian and a thresholding. The images were masked using the measurement of the water depth using acoustic sensors.The images were dewarped using the camera models, obtained via a calibration procedure. The camera models were adjusted in order to be sure that the lines of sight are crossing on the particles. This is a delicate step, but it is necessary in order to avoid biases. Moreover, it can also correct the position of the cameras when it has moved during an experiment.

The images were correlated using the FTEE corre lation algorithm (Jeon et al. (2014)), which is a PIV algorithm adapted to time resolved measurements, and giving velocity field with good trajectories

approximations. The velocity fields (two compo

nents) obtained for each camera were combined Figure 4. Spatio-temporal diagram of the velocity field for Fr = 1.13. The vectors represent the 2D velocity field. The colours represent the horizontal velocity component. (Calluaud and David (2004)) to get the final velocity field (three components). • PTV The images were processed in the same way as for the stereo-PIV. A PTV algorithm was used to follow the particles. The first step was initialized using the velocity fields obtained with stereo-PIV. The particles were tracked using a polynomial predictor. As the time steps are small, and the flow is mostly bidimensional, the particles can be followed on several images, often more than 40. 3 RESULTS 3.1 Eulerian point of view In figures 4, 5, 6 and 7, spatio-temporal diagrams representing the velocity field along a vertical line are shown. One vector over 4 is shown along Y, and one over 50 along t. The horizontal velocity component is illustrated using iso-contours, red for positive values, blue for negative ones. In figure 4, the flow is oscillating very smoothly. When the water level is increasing, the flow is slowing down, in order to keep the flow rate. In figure 5, the behaviour is rather similar. Anyway, when the bore is breaking, the velocity is going in reverse direction. In figure 6, the breaking occurs on two pseudo-periods. The boundary layer is more visible, and as the oscillations are smaller, the deceleration is less clear, except when the bore is arriving. Under the bore, there is a stagnation point near the floor. In figure 7, there is a brutal deceleration when the bore is coming. The breaking zone is large with strong reverse velocities. The boundary layer thickness is increasing all the time. There is a flow reversal near the floor. As the Froude number is increasing, the W velocity component is increasing, especially above the initial water level. 3.2 Lagrangian point of view The equation that governs the movement of a solid particle in a flow is the Maxey-Riley equation. If the particle is small, the Basset history term and the

Figure 5. Spatio-temporal diagram of the velocity field for

Fr = 1.31. The vectors represent the 2D velocity field. The

colours represent the horizontal velocity component.

Figure 6. Spatio-temporal diagram of the velocity field for

Fr = 1.44. The vectors represent the 2D velocity field. The colours represent the horizontal velocity component.

Figure 7. Spatio-temporal diagram of the velocity field for Fr = 1.58. The vectors represent the 2D velocity field. The colours represent the horizontal velocity component.

Faxen corrections can be neglected. The simplified non-dimensionalized equation is given in equation (2).

where Table 3. Characteristic numbers of the particles. Fr d h (m) U f (m/s) R St W 1.13 50 0.118 0.353 0.625 0.0007 8.0·10 −4 1.31 50 0.094 0.665 0.625 0.0017 4.2·10 −4 1.44 50 0.073 0.567 0.625 0.0019 5.0·10 −4 1.58 50 0.143 0.871 0.625 0.0015 1.8·10 −4 1.13 300 0.118 0.353 0.625 0.026 2.9·10 −3 1.31 300 0.094 0.665 0.625 0.061 1.5·10 −3 1.44 300 0.073 0.567 0.625 0.067 1.8·10 −2 1.58 300 0.143 0.871 0.625 0.053 6.4·10 −3 Figure 8. Comparison of the fluid and particles velocities for the case Fr = 1.44 and a = 300 µm. The black vectors is representing the fluid velocity field and the red ones, the particle velocities. St is the Stokes number,W is the settling parameter and R is the density ratio. And a is the particle diameter, µ is the dynamic viscosity of the water. Here the temperature of the water is 21 ◦ C. µ 9.79·10 −4 Pa·s and ρ water 998 kg·m −3 . For our case, the different values are given in table 3. The St and W values are small, hence in most cases, the particles velocities should be close to the fluid flow velocity. Anyway, as these numbers are evaluated for a characteristic velocity and a characteristic length, the values can change inside the flow. The most critical case is Fr = 1.44 and a = 300 µm. Fluid velocities and particle velocities are compared in figure 8. From this figure, the velocity differences are within the experimental uncertainties. Another illustration is given in figure 9, where the blue lines are trajectories obtained tracking the big particles and the black lines are the trajectories integrated using the velocity fields. From this figure, it can be observed that the trajectories are very similar. A visualization of the trajectories is computed using averages over 80 time steps (corresponding to 0.16 s) of the pre-processed images. The result is presented in figures 10, 11, 12 and 13. In the Fr = 1.13 case, the trajectories are very smooth, except in the boundary

Figure 9. Comparison of the fluid and particles trajectories for the case Fr = 1.28 and a = 300 μm. The black lines are obtained through the integration of the velocity fields and the blue lines are obtained tracking the big particles.

layer, near the floor. The boundary layer is thicken ing when the water level is increasing. The Fr = 1.31 case is rather similar, except that the bore front is slightly breaking, with a steeper front, and that the wavelength is shorter. For both cases, when the water level is maximum, the particles are slowing down, and the trajectories are rising. The Fr = 1.44 case is differ ent as the bore is breaking (with a steep front) and as the amplitude of the free surface oscillations is much lower. The particles trajectories are more complex. Some of them seem to be circular, in the boundary layer, where some small vortices can be observed, and near the free surface where some big vortices seem to trap the particles for some time. In the centre of the flow, the trajectories seem to be entangled, as the water depth is the smallest used in the present experiments. The Fr = 1.58 case present some similar characteris tics: the bore is breaking and the flow oscillations are weak. Big structures are visible under the free surface. The trajectories are regular between the boundary layer and the structures under the free surface.

A way to analyze the flow a bit further is to investigate the stability of the trajectories. The trajectory of a point at X_0 at time t_0, is given by equation 6.

$\varphi_{t_0}^{t_0+T}$ is the flow map. The evolution of two particles at positions X_0 and $X_0 + \delta X_0$, initially very close ($||\delta X_0||$ very small), is given by the positions $X(t_0 + T)$ and $X(t_0 + T) + \delta X(t_0 + T)$. $\delta X(t_0 + T)$ is given by equation 7.

F is the flow map gradient. It is computed by computing the trajectories of 4 particles and using finite differences to compute the gradient. After an observation time T, the distance $l = ||\delta X||$ between the particles is given in equation (8).

where $C=F^* F$ is the Cauchy-Green tensor. The maximum distance l_{max} is given in equation (9). where λ_{max} is the greatest eigenvalue of the CauchyGreen tensor (Shadden et al. (2005)). The FTLE (Finite Time Lyapunov Exponent) [illegible](T) is given in equation (10). In principle, the highest particle separation in flow is exponential. In this case, the FTLE is independent of the integration time T. In the case of a simple shear, the separation is linear, hence the FTLE is decreasing with the integration time T. The FTLE fields are given in figures 14, 15, 16 and 17. The computation time is $T = 0.1$ s. Eight fields have been stuck together to give an impression of the time evolution of the fields. The red values represent strong trajectories unstabilities, while blue values gives stable trajectories. Near the free surface, there is always strong unstabilities due to the velocity jump. Near the floor, FTLE fields are green or red, showing the presence of strong shear. Very strong shear occurs also near the breaking of the bore. In figure 14, except in the boundary layer, the trajectories are very stable. There is only a slight augmentation of the distance between two adjacent trajectories under the bore but they go back to their original configuration afterwards. In figure 15, there are more unstable trajectories. The highest values are situated where the bore is breaking.The boundary layer is thicker than for Fr = 1.13. In the middle, there are still regions where the trajectories are stable. There is a thick layer also under the free surface. In figure 16, there are no more stable trajectories.As the water depth is the lowest, the boundary layer on the floor and the layer under the free surface are meeting. There is still a strong signature of the bore breaking. The last case (figure 17), there is even more energy in the bore. The breaking is strongly emphasized and its effects last for a long time. The unstable layer under the free surface goes deep in the water. The floor boundary layer is thick, but as the water depth is high, is doesn't reach the upper layer. Stable trajectories are going through the channel between the two layers. 4 CONCLUSION The study of moving bores in laboratories is complementary to in-situ measurement campaigns. The big advantage is that it is possible to use very sophisticated measurement techniques to scrutinize all the specificities of this kind of unsteady flow. In this paper, stereo-PIV measurements coupled with PTV measurements have been used to show the different behaviours of the flow under the bore. For these study cases, no real local differences have been shown between the flow and the particle velocities for a range of particle diameters included between 50 to 300 µm. Moreover, the study of the trajectories and their stability gives new insights into the way that the flow is structured. The regions of potential high impact on the sediment transport appear clearly.The Lagrangian analysis

Figure 10. Visualizations of the trajectories under the bore front for Fr = 1.13. Each line has a duration of 0.16 s.

Figure 11. Visualizations of the trajectories under the bore front for Fr = 1.31. Each line has a duration of 0.16 s.

Figure 12. Visualizations of the trajectories under the bore front for Fr = 1.44. Each line has a duration of 0.16 s.

Figure 13. Visualizations of the trajectories under the bore front for Fr = 1.58. Each line has a duration of 0.16 s.

Figure 14. FTLE field under the bore front for Fr = 1.13. The integration time is T = 0.1 s. The highest values correspond to

the red color, and the lowest to the blue color.

Figure 15. FTLE field under the bore front for Fr = 1.31. The integration time is T = 0.1 s. The highest values correspond to

the red color, and the lowest to the blue color.

Figure 16. FTLE field under the bore front for Fr = 1.44. The integration time is T = 0.1 s. The highest values correspond to

the red color, and the lowest to the blue color.

Figure 17. FTLE field under the bore front for Fr = 1.58. The integration time is T = 0.1 s. The highest values correspond to

the red color, and the lowest to the blue color.

has highlighted different trajectories dependant of the Froude number and the flow structures close to the bottom. When the bore is undular, the trajectories are very stable. The only effect is a slight divergence of the trajectories under the bore. Anyway, it can have an effect on the structuration of the bed, but in the present experiment, the sediments were already in the flow.

For higher Froude numbers, the impact of the bore on the flow is more dramatic. Strong vortex structures appear under the free surface, and the boundary layer is thickened, leading to the birth of new structures in the flow. These structures should have a great impact on the resuspension of the sediment. The erosion at the bottom will be the subject of further studies.

ACKNOWLEDGEMENT

The authors would like to acknowledge the French

National Agency for their financial support in the

Project BLANC MASCARET 10-BLAN-0911, the

FEDER and the Region Poitou-Charentes.

Calluaud, D. & L. David (2004). Stereoscopic particle image

velocimetry measurements of the flow around a surface

mounted block. Experiment in Fluids 36(1), 53–61.

Chanson, H. (2005). Physical modelling of the flow field in an

undular tidal bore. Journal of Hydraulics Research 43(3),

234–244.

Chanson, H. & K. K. Tan (2010). Turbulent mixing of parti

cles under tidal bores: an experimental analysis. Journal

of Hydraulics Research 48(5), 641–649. Chen, S. (2003). Tidal bore in the north branch of the changjiang estuary on erosion and sedimentation. In Proc. International Conference on Estuaries and Coasts 1, Hangzhou, China, pp. 233–239. David, L., L. Chatellier, D. Calluaud, Y. Jeon, G. Rousseaux, & L. Thomas (2014b). Tr-piv measurements in open channel flow for the analysis of undular tidal bores. In 17th International Symposium on Applications of Laser Techniques to Fluid Mechanics, Lisbon, Portugal. David, L., B. Lebon, L. Chatellier, & D. Calluaud (2014a). Influence of initial flow boundary conditions on undular tidal bores. In Proc. 3rd IAHR Europe Congress, Porto, Portugal. Fan, D., G. Cai, S. Shang, Y. Wu, Y. Zhang, & L. Gao (2012). Sedimentation processes and sedimentary characteristics of tidal bores along the north bank of the qiantang estuary. Chinese Science Bulletin 57, 1478–1589. Jeon, Y. J., L. Chatellier, & L. David (2014). Fluid trajectory evaluation based on an ensemble averaged cross-correlation in time resolved piv. Experiment in Fluids 55:7, 1766. Kjerfve, B. & H. Ferreira (1993). Tidal bores: first ever measurements.

Journal of the Brazilian Association for the Advancement of Science 45, 135–138. Shadden, S. C., F. Lekien, & J. E. Marsden (2005). Definition and properties of lagrangian coherent structures from finite-time lyapunov exponents in two-dimensional aperiodicflows. Physica D: Nonlinear Phenomena 212(34), 271–304. Simon, B., P. Lubin, S. Glockner, & H. Chanson (2011). Three-dimensional numerical simulation of the hydrodynamics generated by a weak breaking tidal bore. In Proc. 34th IAHR World Congress, Brisbane, Australia. Wolanski, E., D. Williams, S. Spagnol, & H. Chanson (2004). Undular tidal bore dynamics in the daily estuary. In Estuarine, Coastal and Shelf Science 60, Northern Australia, pp. 629–636. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Effect of hydrodynamics factors on flocculation processes in estuaries

A. Mhashhash, B. Bockelmann-Evans & S. Pan

School of Engineering, Cardiff University, UK

ABSTRACT: Cohesive sediment flocculates under certain conditions to form flocs which are larger than

individual particles and less dense. The flocculation has an important role in sediment transport processes

of settling, deposition and erosion. In this study, well controlled laboratory experiments were performed to

investigate the effect of turbulence on floc size and settling velocity. Experimental research was conducted in

a 1 L glass beaker of 11 cm diameter using suspended sediment samples taken from the Severn Estuary. A PIV

system and image processing routine were used to measure the floc size distribution and settling velocity, as well

as the flocs density. This study found that turbulent shear stress in a range from 0.57 to 8.5 N/m 2 would cause

a breakdown in floc structure. The settling velocity of the samples was found to range from 0.4 to 1.4 mm/s.

The average settling velocity increased with the increase of the turbulent shear stress up to the maximum value

(1.1 mm/s), and then decreased. It was also found that the relationship between floc size and turbulence were independent of the history of the floc formation.

1 INTRODUCTION

Cohesive sediments are regarded as one of the most important features of estuaries around the world. The sediment size of the cohesive sediment is normally in a range of 2–63 μm. Under certain conditions, these sediments flocculate to form large aggregates, namely flocs, which are larger than individual par ticles but less dense. This flocculation phenomenon has a strong influence on the sediment transport pro cesses of deposition, erosion and settling (Fennessy et al. 1994).

In general, flocs were classified as two types, namely, microflocs and macroflocs, whereas cohe sive sediment flocculates to form small microflocs first, and then macroflocs by combining the microflocs (Eisma 1986). Microflocs can be classified as those aggregates which do not exceed a spherically equiv alent diameter of 100 μm (Lafite 2001) and have a settling velocity of less than 1 mm/s. The state of microflocs continually changes in response to the hydrodynamic, and the physico-chemical and environ mental conditions. These microflocs can develop into large, but low dense flocs called macroflocs which behave very differently. Macroflocs have a diameter larger than 100 μm and a settling velocity between 1–15 mm/s (Fennessy et al. 1994; Manning and Dyer 1999; Manning 2001; Manning et al. 2013).

Flocculation of the fine sediments can occur with two different mechanisms: one is to bring the parti cles to direct contact by turbulence; and the other is to stick the flocs together by electrostatic charging e.g. organic matter content and salinity (Dyer and Man ning 1999). The flocculation process mainly occurs in the very low salinity region (between 1 and 2.5 ppt) (Wollast 1988), and it is affected by hydrodynamic changes which can alter the suspended sediment particle by modifying its effective particle size, shape, porosity, density and composition. Numerous studies have been carried out to investigate the flocculation phenomenon in the laboratory, such as (Serra et al. 1997; Manning and Dyer 1999; Mikes et al. 2004; Maggi 2005) and in situ (Fennessy et al. 1994; Syvitski et al. 1995; van Leussen and Cornelisse 1996). Due to the complexity of the natural system, many simplifications are made in laboratory studies in order to control the different variable parameters in the flocculation process. Natural flocculation processes are difficult to be reproduced in laboratory experiments due to the complexity of processes involved. However, the laboratory experiments are valuable for investigating systematically the effect of specific parameters such as salinity, suspended sediment concentration and turbulence under controlled conditions (Manning 2004b; Manning et al. 2004; Mikes et al. 2004). One of the main parameters for studying the flocculation process in laboratory experiments is the turbulence, which is used to create conditions as close to the natural estuarine environment as possible. Examining floc distribution under changing turbulent shear rate may lead to multiple cycles of growth, break up, and regrowth that produce conditions are, to some extent, similar to situ measurement. Four different devices, namely the jar test (Mikes et al. 2004), annular flume (Dyer and Manning 1999), sedimentation column and turbulence grid (Maggi 2005), and the Couette device with a video camera system (Serra et al. 1997; Serra and Casamitjana 1998) have been used for generating turbulence and floccu lation in laboratory experiments. The development of the latter technology (video technology) has allowed obtaining both floc size and settling velocity spectra.

In this study, a laboratory video analysis method was developed to measure the size of the flocs. This instru mental set-up requires little equipment and is easy to implement in the laboratory. It consists of a glass bowl, CCD camera and variable speed agitator control for the turbulent shear stress inside the bowl.The floc size can then be measured under varying turbulence conditions.

The aim of this study is to experimentally determine the effect of turbulence as one of the key hydrody namic parameters, on the flocculation phenomenon.

Floc size and settling velocities are measured during these experiments.

2 MATERIALS AND METHODS

2.1 Overview of study area

The Severn Estuary, located between South East Wales and South West England, is the largest tidal estuary in the UK and has the third highest tidal range in the world with a spring tide of up to 14.7 m (Kadiri 2014). The estuary generates high currents that exceed 3 m/s (Gao et al. 2011). The river system has a total catchment area of approximately 25,000 km 2 (Jonas and Millward 2010) and the estuary has a total channel length of 137 km. The major tributaries of the Severn are the Usk, Wye (on the Welsh side) and the Stour (on the English side). The tidal range varies signifi cantly along the estuary and over time. The average spring and neap tidal ranges are 12.3 m and 6.5 m, respectively (Kirby 2010). The annual suspended sedi ment load has been approximated at 1.6×10^9 kg/year; nearly 1.25×10^9 kg/year of which is discharged from the riversWye,Avon and Severn (McLaren et al. 1993). Flocculation phenomenon was studied in the lab oratory using suspended sediment from the Severn Estuary. Samples were collected from 'Slipway' in the proximity of the New Severn Bridge as shown in Figure 1. Data acquisition took place on 1 June 2014, sampling at around the high tide mark of 9.5 m.

2.2 Instrumentation

Flocculation experiments were conducted in 1 L glass beaker of 11 cm diameter. It was equipped with a vari able speed agitator to control turbulence of the flow inside the beaker. A settling column with a diameter of 5 cm and a height of 40 cm was used to measure the flocs' settling velocities. Flocs were introduced from the top of the settling column filled with water, where the falling flocs were filmed using a PIV sys

tem as shown in Figure 2. The PIV system consists

of a backlight which is positioned opposite the CCD

camera to provide a uniform black background upon

which particles appear as white, the CCD camera with

1392 × 1040 pixel sensitivity, focal length, f, of 9 mm
Figure 1. Map of Severn Estuary showing the location of sam-pling sites, Slipway ($2^{\circ} 30' 00.67''$ N,$51^{\circ} 42' 52.12''$ W). Figure 2. Schematic diagram of PIV system, settling column, sample bowl and stirrer. and a maximum frequency of 30 fps (Δt = 1/30 s), a Polytec BVS-11 Wotan flash stroboscope and trigger box, fibre optic cable and linelight. 2.3 Instrument calibration Instrument calibration for the flow velocity is necessary to quantify the hydrodynamic shear stress present inside the beaker during the experiment. The angular velocities (ω) of the agitator were set at 37, 50, 70, 90 and 110 rpm. The turbulence is normally obtained by three velocity components but the radial velocity (w') was found too small. Therefore, at each rotational speed, the turbulent kinetic energy (K) was measured

Table 1. Shear flow parameters with respect to different angular velocities.

Angular velocity	Flow velocity Measured	Flow velocity Calculated	Turbulent shear stress
rpm	m/s	m/s	N/m^2
37	0.10	0.08	0.57
50	0.14	0.14	1.70
70	0.20	0.22	3.80
90	0.26	0.21	6.00
110	0.32	0.29	8.50

by considering the tangential and vertical velocity

fluctuations (u', v'). The fluctuation velocity can be measured by using the PIV equipment. Since turbulent kinetic energy is defined according to the following equation:

The equation (2) is used to determine the turbulent shear stress (T_s) in N/m^2 (Manning 2004a):

where ρ_w is the water density and which was assumed to be 1000 kg/m^3.

The flow velocity was also calculated theoretically by conversion of the angular velocity into the linear velocity.

where, D is the diameter of the beaker (mm), ω is the angular velocity (rpm) and u is a flow velocity (m/s).

Table 3 presents the calculated turbulent shear stresses that correspond to all of the rotational speeds and the experiment (measured) and theoretical (cal culated) average flow velocity values. Comparison between the theoretical and experimental flow veloc ity is shown in Figure 3. The theoretical and measured (using PIV) flow velocities were found to correlate reasonably well with a R^2 value of 0.98. This result confirms the accuracy of the PIV camera and the suit ability of this method to this novel type of application as also applied by Maggi (2005).

2.4 Experimental procedures

This study focuses on the influence of the turbulent shear stress on the floc size and settling velocity.

The experiments were carried out in two main steps:

The first step is to apply the highest tested shear stress

of 60 N/m^2 to break down any potential macroflocs in

suspension as the initial state; the second step, which

is the main testing part, is to run the agitator at a series Figure 3. Comparison between experimental and theoretical flow velocities. of rotational speeds to generate the desired turbulent shear stresses, ranging from 0.57 N/m^2 to 8.55 N/m^2 for a duration of 120 min as suggested by (Mikes et al. 2004; Verney et al. 2009). This is a significant time period for flocculation to occur in natural water bodies (Le Hir et al. 2001), where the average floc size no longer changes after this time. During the experiment, a series of images is recorded from the PIV system at different time steps in order to calculate the floc size distributions over the investigated period. To investigate the effect of turbulent shear stress variation on flocs size and settling velocity, 5 laboratory experiments with turbulent shear stresses of 0.57, 1.7, 3.8, 6 and 8.55 N/m^2 were conducted for the same suspended sediment concentration and salinity, c = 100 mg/l and S = 2.5 ppt, respectively. These values of salinity and sediment concentration were chosen in this study because they were considered to represent the optimal conditions for flocculation processes. The organic carbon represents 2%. For each test, 10 ml of flocculated sample was then introduced from the top of the settling column.After introduction of the sample into the water column, the flocs were allowed to settle by gravity over a distance of approximately 13 cm prior to switching the camera on and in order to allow the damping out of any activity from the introduction method. Another specific set of experiments was conducted in this study to investigate the effect of turbulence over time on the particle size. For this scenario only one parameter was varied at a time, while the other was kept constant. The test was undertaken at salinity of 2.5 ppt, sediment concentration of 100 mg/l and the lowest turbulent shear stress of 0.57 N/m^2 for 2 hours, allowing the floc grow from the initial state to equilibrium size to an equilibrium state (Le Hir et al. 2001; Mikes et al. 2004). Then the turbulent shear stress was increased every hour until reaching 8.55 N/m^2 .The test was undertaken at salinity of 2.5 ppt, sediment con centration of 100 mg/l and the lowest turbulent shear stress of 0.57 N/m 2 for 2 hours. This time is important to allow the floc grow from the initial state to equi librium size, where the average floc size no longer changed with time (Le Hir et al. 2001; Mikes et al. 2004) stopped at each turbulent shear stress. The agi tator was for approximately 3–5 seconds in order to take some images for analysis of the size of the flocs.

2.5 Image analysis

The floc size distribution and settling velocity were both obtained from floc image recording and process ing. The main five steps in the image processing are needed: (1) selecting the flocs manually at the start and at the end of the sequence by opening images using image editor and paint program; (2) enhancing back ground (brightness and contrast); (3) removing any noise to make sure the flocs appear in all of the sequen tial images by opening images using an image editor and the paint program; (4) removing all flocs which are touching the image boundary and are not in focus; and (5) calculating the features of flocs including: sectional area, location and circularity by using the “ImageJ” software. As this method is interactive there is very low risk of errors being made in the determination of the floc paths.

ImageJ was capable of detecting particles larger

than 70 μm below this limit the pixel resolution of

the floc measurement is not consistent and hence the

smallest microflocs are not accounted for the descrip

tion of the floc population during the experiment.

Floc size was obtained using the contrast between

the dark background and the white silhouettes of the

floc. The surface equivalent diameter d was calculated

by converted particle area (A) into equivalent circular

diameter (Flory et al. 2004; Mikes et al. 2004; Verney

et al. 2009) as:

3 RESULTS AND DISCUSSION

This section describes the laboratory experiments

results of the influence of the turbulent shear stress on

floc size distribution (FSD) and their settling velocity.

3.1 Effect of turbulence on FSD

The floc size distributions under different shear veloc

ities are shown in Figure 4.

Table 2 shows the size distributions in eight bands.

Band 1 represents flocs with a size of less than 100 μm,

whilst band 8 represents flocs with a size bigger than

700 μm. Other sizes of the flocs are evenly repre

sented by bands 2 to 7. For the smallest shear stress

(0.57 N/m 2), it can be seen that no particles smaller

Table 2. The definition of floc size band. Size band 1 2 3 4 5 6 7 8 Floc size < 100200300400500600> (μm) 100 200 300 400 500 600 700 700 than 100 μm are detected and there is no detection of particles larger than 500 μm. Whereas, for the shear stress at 8.5 N/m 2 , 50 % of the floc area have particles with a size of less than 100 μm and there is no particle detected larger than 400 μm.This confirms that higher shear stresses lead to break down of macroflocs into microflocs. The floc structure are shown in Figure 5, by the SEM photographs at η= 0.57 N/m 2 for sediment concentration of 100 mg/l and salinity 2.5 ppt. The irregular shape floc can be clearly seen in Figure 5. Although, it cannot be directly measured of the floc structure using PIV camera, the fractal dimension of flocs can be determined theoretically using Winter-werp model (Winterwerp 1999). This model was developed based on field and laboratory data, as shown in Figure 6. The experiment data of settling velocity and floc size at different turbulent shear stresses was plotted withWinterwerp model in Figure 6. It is observed that the data match the Winterwerp model adequately. The overall trend of the experiment data points seems slightly steeper than the fit with nf = 2. However, when the individual data set are studied, the slope agree better with nf = 2 for turbulent shear stress less than 6 N/m 2 . It is important to work with nf value as the density is more realistic than the stokes law. As in the stokes law the density calculated is based on the assumption of the flocs have a spherical diameter. By knowing nf value from this chart, it will be easy to calculate the floc density from theoretical equation: where ρ f is the floc density, ρ w is the water density, ρ s is the mud density, d is the equivalent spherical diameter, d i is the diameter of the primary particle and nf is the fractal dimension. 3.2 Effect of turbulence change over time on FSD Evidently, turbulence plays an important role on flocculation mechanism. In natural estuarine waters turbulence varies during the tidal cycle (Zhu et al. 2015). This specific scenario was explored in this as the effect of turbulence on FSD over time. An increase of the turbulent shear stress from 0.57 N/m 2 to 8.55 N/m 2 was considered with a constant concentration (c) of 100 mg/l and salinity (S) of 2.5 ppt. Figure 7 represents the maximum floc size for suspended sediment concentration of 100 mg/l and salinity of 2.5 ppt, as a function of the turbulent shear

Figure 4. Floc size distribution under different turbulent

shear stress including the standard deviation between two runs.

Figure 5. SEM photographs of floc from the experiment at

turbulent shear stress of 0.57 N/m^2 .

rate calculated to the corresponding stirring velocity.

It is apparent that the turbulent shear stresses rang

ing from 0.57 N/m^2 to 8.5 N/m^2 cause a breakdown in

floc structure rather than enhancing the flocculation

processes, causing a decrease of the floc diameter. The

floc strength was addressed indirectly when we looked

at the effect of maximum floc under different turbulent

shear stress as a PIV was incapable of taking directly

measurement of the floc strength. This observation is

a similar to that given by (Manning and Dyer 1999)

who found a decrease in floc size with an increase of

the shear velocity of up to 0.3 N/m^2 in their laboratory

flume experiments with sediment samples from the

Tamer Estuary and sediment concentrations ranging

from 80 mg/l to 200 mg/l.

The results of two tests with different turbulent

shear stresses on samples with maximum size flocs

are shown in Figure 8. Figure 6. Relation between settling velocity and floc size. Figure 7. Maximum Floc size changing over time under different turbulent shear stress. Two sets of experiments were carried out. In the first set, 5 different turbulent shear stresses were tested in at 0.57, 1.7, 3.8,6 and 8.5 N/m^2), which represent the variations in the maximum floc size over fixed turbulent shear stress. Whereas, in the second set, only one test was run starting at a turbulence of 0.57 N/m^2 for 2 hours initially and then turbulence was increased to 1.7, 3.8, 6 and 8.5 N/m 2 at every 1 hour interval which represents the relationship between the maximum floc size and the variation of the turbulent shear stress over time. Figure 8 clearly shows a large floc size occurred at the lowest speed (0.57 N/m 2). It was found that when turbulence was increased the flocs broke apart which was reflected in a decrease of the maximum diameter from 865 μm at η= 0.57 N/m 2 to 280 μm at η= 8.5 N/m 2 , of nearly 67.6%.

Figure 8. Maximum floc size for: a) fixed agitator speed and b) varying agitator speed over time. The range shows the standard deviation between two runs.

The maximum floc size was found to be the similar with very little differences. This means that the floc size becomes independent to the turbulent shear stress once it is formed during the initial stage.

3.3 The influence of turbulence on settling velocity

The settling velocities against floc size for different turbulent shear stresses are displayed in Figure 9. This figure shows that the settling velocity changes from 0.4 to 1.4 mm/s. The settling velocity changes with both parameters: floc size and turbulence. For a constant shear stress, the settling velocity increased with particle size. The settling velocity increased by almost 45.5% from 0.6 m/s 1 at turbulent shear stress of 0.57 N/m 2 to reach a maximum value (1.1 mm/s) at η= 6 N/m 2 , followed by a 27% decrease (0.8 mm/s) at a maximum value of η (8.5 N/m 2).

Pejrup and Mikkelsen (2010) showed that the mean settling velocity increases as a function of velocity gra dient up to a maximum value of approximately $8.5\ s^{-1}$,

and then starts to decrease. This result was based on

data collected by (Pejrup et al. 1997) from Rømø Dyb

where the suspended sediment concentration varied

between 20 and 200 mg/l. It was found in this study that

for a certain range of settling velocities the increasing

floc size can be described by the regression equation

in the form of $y = ax + b$. The value for a and b are

constant and are found to be dependent on the tur

bulence. The regression fitness coefficient R^2 ranges

from 0.9 to 0.99 for the entire range of turbulence. The

regression constants and R^2 are listed in Table 3.

4 CONCLUSIONS

The potential impacts of the turbulent shear stresses

on the floc size and settling velocity were assessed Figure 9. Settling velocity with floc size and for varying turbulent shear stress.The range shows the standard deviation between two runs. Table 3. Settling velocity equations as a function of floc size developed from the experiment. Regression Constants Turbulence (N/m^2) a b R^2 0.57 0.0016 0.36 0.99 1.7 0.0025 0.20 0.93 3.8 0.0018 0.56 0.99 6.0 0.0018 0.70 0.91 8.5 0.0020 0.46 0.96 in this study using suspended sediment samples taken from the Severn Estuary with well-controlled laboratory experiments. The results demonstrated that for turbulent shear stresses ranging from 0.57 to 8.5 N/m^2, a breakdown in floc structure was occurring by nearly 67.6%. The results clearly illustrate that the settling velocity increases by almost 45.5% from (0.6 mm/s) at turbulent shear stress of 0.57 N/m^2 to reach a maximum value of 1.1 mm/s at turbulent shear stress of 6 N/m, followed by a 27% decrease to 0.8 mm/s at the maximum turbulent shear stress of 8.5 N/m^2 in the experiments. Additionally, it was found that the relationship between floc size and turbulent shear stresses is independent of the history of the floc formation. Deposition and resuspension processes in estuaries are strongly dependent on floc size and settling velocity, which depend mainly on the turbulent shear stress. Therefore, the results obtained from this study can be implemented into the numerical models so that flocculation mechanisms (floc size and settling velocity) can be considered more realistically for the field scale, and for the direct impact on morphological and water quality processes.

Representation of these processes in hydrodynamic estuarine models will lead to a better understanding of how to best manage estuarine and coastal waters under future stresses like climate change.

Future work will include applying and refining a numerical model does not have a flocculation pro cesses to include the more complex settling velocity function taking in account density and floc size for dif ferent salinity, suspended sediment concentration and turbulent range. This will be the first useful step.

Dyer, K. R. and Manning, A. J. 1999. Observation of the size, settling velocity and effective density of flocs, and their fractal dimensions. Journal of Sea Research 41(1-2), pp. 87-95.

Eisma, D. 1986. Flocculation and de-flocculation of sus pended matter in estuaries. Netherlands Journal of Sea Research 20(2-3), pp. 183-199.

Fennessy, M. J. et al. 1994. inssev: An instrument to mea sure the size and settling velocity of flocs in situ. Marine Geology 117(1-4), pp. 107-117.

Flory, E. N. et al. 2004.The relationship between floc area and

backscatter during a spring phytoplankton bloom. Deep

Sea Research Part I: Oceanographic Research Papers

51(2), pp. 213–223.

Gao, G. et al. 2011. Numerical Modelling Sediment-Bacteria

Interaction Processes in the Severn Estuary. Journal of

Water Resource and Protection 3(1), pp. 22–31.

Jonas, P. J. C. and Millward, G. E. 2010. Metals and nutrients

in the Severn Estuary and Bristol Channel: Contempo

rary inputs and distributions. Marine Pollution Bulletin

61(1–3), pp. 52–67.

Kadiri, M. B.-E., B.; Zhou, J.;Falconer,R. 2014.Water Quality

Impacts of A Tidal Barrage In the Severn Estuary,UK. In:

10th ISE 2014. Trondheim, Norway.

Kirby, R. 2010. Distribution, transport and exchanges of fine

sediment, with tidal power implications: Severn Estuary,

UK. Marine Pollution Bulletin 61(1–3), pp. 21–36.

Lafite, R. 2001. Impact de la dynamique tidale sur le transfert

de sédiments fins. Université de Rouen.

Le Hir, P. et al. 2001. Fine sediment transport and accumula

tions at the mouth of the Seine estuary (France). Estuaries

24(6 B), pp. 950–963.

Maggi, F. 2005. Flocculation dynamics of cohesive sediment.

University of Technology, Delft.

Manning, A. J. 2001. Study of the effect of turbulence on the

properties of flocculated mud University of Plymouth.

Manning,A. J. 2004a. Observations of the Properties of Floc

culated Cohesive Sediment in Three Western European

Estuaries. Journal of Coastal Research, pp. 70–81. Manning, A. J. 2004b. The Observed Effects of Turbulence on Estuarine Flocculation. Journal of Coastal Research, pp. 90–104. Manning, A. J. and Dyer, K. R. 1999. A laboratory examination of floc characteristics with regard to turbulent shearing. Marine Geology 160(1–2), pp. 147–170. Manning, A. J. et al. 2004. Flocculation measured by video based instruments in the gironde estuary during the European commission SWAMIEE project. Journal of Coastal Research (SPEC. ISS. 41), pp. 58–69. Manning, A. J. et al. 2013. Flocculation Dynamics of Mud: Sand Mixed Suspensions. McLaren, P. et al. 1993. Sediment dynamics of the Severn Estuary and inner Bristol Channel. Journal Geological Society (London) 150(3), pp. 589–603. Mikes, D. et al. 2004. “Controlling factors in estuarine flocculation processes: experimental results with material from the Seine Estuary, Northwestern France.” Journal of coastal research 41, pp. 82–89. Pejrup, M. et al. 1997.A fine-grained sediment budget for the Sylt-Rømø tidal basin. Helgoländer Meeresuntersuchungen 51, pp. pp. 253–268. Pejrup, M. and Mikkelsen, O.A. 2010. Factors controlling the field settling velocity of cohesive sediment in estuaries. Estuarine, Coastal and Shelf Science 87(2), pp. 177–185. Serra, T. and Casamitjana, X. 1998. Structure of the Aggregates During the Process of Aggregation and Breakup Under a Shear Flow. Journal of Colloid and Interface Science 206(2), pp. 505–511. Serra, T. et al. 1997. Aggregation and Breakup of Particles in a Shear Flow. Journal of Colloid and Interface Science 187(2), pp. 466–473. Syvitski, J. P. M. et al. 1995. In-situ characteristics of particles settling within a deep-water estuary. Deep Sea Research Part II: Topical Studies in Oceanography 42(1), pp. 223–256. van Leussen, W. and Cornelisse, J. M. 1996. The underwater video system VIS. Journal of Sea Research 36(1–2), pp. 77–81. Verney, R. et al. 2009. Flocculation potential of estuarine particles: The importance of environmental factors and of the spatial and seasonal variability of suspended particulate matter. Estuaries and Coasts 32(4), pp. 678–693. Wollast, R. 1988. The Scheldt Estuary. W. Salomon, B.L. Bayne, E.K. Duursma, U. Förstner (Eds.), Pollution of the North Sea, an assessment, Springer-Verlag, pp. 183–193. Zhu, Z. et al. 2015. Fractal Dimension of Cohesive Sediment Flocs at Steady State under Seven Shear Flow Conditions. Water 7(8), p. 4385. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Lack of scale separation in granular flows driven by gravity

G. Rossi & A. Armanini

Department of Civil, Environmental and Mechanical Engineering, University of Trento, Trento, Italy

ABSTRACT: Since about three decades, granular flows have been studied through the kinetic theories, origi

nally derived for gases: the reason is a certain similarity between granular flows and gases, given that granular

collisions can be considered an equivalent of the molecules collisions. However there are important differences

between the two applications: in molecular gases the microscale, represented by the mean molecular free paths,

is very much smaller than the macroscale, according to which gradients change (Goldhirsch 2003, Goldhirsch

2008), while in granular flows driven by gravity the dimension of a single particle becomes comparable to that

of the control volume. Statistically speaking, the system is no more ergodic: the averages of a process parameter

done over time, over space and over a statistical ensemble do not coincide. For this reason we believe that, instead

of the usual ensemble average, a different type of average is needed. A deep analysis of the implications due to

the averaging processes applied at different scale must be taken into account. In particular, an intermediate scale

will be solved applying a type of average, which accounts for the fluctuations of the number density and leads

to additional diffusive terms. The closure relation of the small scale currently adopted in the kinetic theories

will be extended to the intermediate scale by scaling the diffusive coefficient according to the square root of

the granular temperature and the flow depth h or the particles distance from the wall (instead of the particle

diameter d, used for the small scale). A proper definition of the diffusive coefficient will be provided through

an intensive experimental investigation.

1 INTRODUCTION

Environmental granular flows present a very complex

mechanical behaviour, involving more than one phase

and different types of regimes.

The first attempt to model their mechanics through a

mathematical and physical frame was done by Bagnold

(Bagnold 1954): he introduced the concept of disper

sive pressure, assuming that the granular pressure is

generated within the flow by collisions among parti

cles. According to Bagnold, this process is dependent

on the concentration of the solid phase: in particu

lar, for low concentration, the flow is characterized by

strong particle velocities, by instantaneous collisions

and by collisional stresses proportional to the square

of the velocity gradient. This is the regime to which

we refer to as collisional.

In the last few decades this type of behaviour

has been compared to that of molecular gases and the kinetic theories have been adopted to study the mechanics of granular flows (Jenkins & Savage 1983, Jenkins & Richman 1985, Lun 1991).

The collisions among particles have been considered an equivalent of those between molecules in gases and the pressure within the granular flow is due to these contacts. The single particle characteristics are described by the distribution function, which recalls the Gaussian distribution and represents what percentage of particles are in a certain zone of the control volume within a certain range of velocity; only binary and instantaneous collisions are expected to occur and their distribution functions are correlated through a function g(r), called radial distribution function. Kinetic theories are based on the Boltzmann equation, which expresses the conservation laws for the properties of the flow. It is based on the average done over all the particles of the control volume. It is worth to notice that some hypothesis behind the kinetic theories do not hold for the granular case: i) the dimension of the particles are now comparable to that of the control volume, so that the strong scale separation is lost, ii) the system is not ergodic as for gases, and the averaging process becomes crucial. The aim of this paper is to analyze the differences induced in the equations derived from the kinetic theories, according to different types of average, and individuate the most appropriate averaging process. Contrary to gases, in granular flows, where the grain size is comparable to the dimension of the control volume, the velocity and the concentration of particles come out to be correlated, so that the averaged product of their fluctuating components is not zero. For this reason the kinetic equations usually applied to granular flows, need further terms. With reference to the continuity equation, we will derive a diffusive term through a Reynolds average decomposition process. The experimental investigation will support the mathematical frame.

2 SCALE SEPARATION AND ERGODICITY

One of the fundamental hypothesis behind the kinetic theories is that the control volume analyzed is taken large enough to contain a very high number of parti cles ($n \to \infty$), but small enough so that compared to the macroscopic dimensions it may be considered a point (Huang 1928). This implies a strong scale sepa ration: the molecules are of an infinitesimal size, so that the scale associated to their mean free path is very much smaller than the scale at which gradients of physical properties occur. If we consider a gradient of the temperature in a gas, even if it is quite large as 200 K/cm, the difference of temperature between two adjacent molecules is negligible (Goldhirsch 2008).
This is a very clear example of the strong scale separa tion, which implies that a macroscopic gradient does not influence what characterizes the micro-scale.
The choice of a control volume that respects these hypotheses is clearly not possible in granular flows, since the grain size scale is similar to that of the gra dients. The dimension of the control volume becomes comparable to that of the particles and a sharp dis tinction between the microand macro-scale is not applicable.
One of the consequences of this aspect, is that the system is not ergodic, as it is for gases (Jaeger H.M.

& R.P. 2008). The ergodicity property means that the average of a process variable made over time, over space and over the whole statistical ensemble coin cides. Due to the lack of strong scale separation, the number of particles in the control volume is not so high to be considered constant, since a few particles may change significantly the concentration.This aspect and the presence of gradients at the particles size may dif ferentiate the average, according as it is done over time, space or statistical ensemble. For this reason, the ergodicity, applicable in the case of gases, is no more a certain property in granular flows.

2.1 Main scales present in literature

In granular flows, we may consider the presence of different scales. The smallest length scale is that pro portional to the particles diameter; it is currently analyzed through the Boltzmann equations, adopting the ensemble average and considering the particles small enough so that the control volume contains an infinite number of them.This approach is derived from the kinetic theories (Jenkins & Savage 1983, Jenkins & Richman 1985, Lun 1991).

However, while for gases this is a realistic hypoth esis, for granular flows, where the particles have a dimension comparable to that of the control volume, this scale turns into an intermediate scale. In fact, the hypotheses behind the Boltzmann procedure are no more true and the scale at which we need to average is greater than the microscopic level, but smaller than the scale related to the external boundaries, which in presence of an interstitial fluid corresponds to that of large turbulence eddies. Indeed, the last scale we may distinguish is that related to the Reynolds stresses, and this is the largest we may consider. For dry granular material, we have no interstitial fluid and this last scale is not present. For this reason, we may expect that the process can be studied considering an intermediate scale, but the two scales most often are not easily separable. In the following section, we analyze the consequences of this lack of scale separation. Notice that the notation usually adopted in the kinetic theories for the velocity is c. In this context, in order to avoid confusion between the concentration (usually represented by c in hydraulics) and the velocity we will indicate the velocity vector with u and the concentration with c. 2.2 Implications of the scale separation on the kinetic theories equations Analyzing the definition of the number density (which represents the volume fraction concentration), we can say that n = f (r, t) (where r indicates the vector of the position), i.e. it doesn't depend on the velocity because by definition it is the integral over all the velocities of the distribution function: Since the ensemble average of a general property of the granular gas is defined as follows: we can conclude that if we have an ensemble average of a property multiplied by n, this one could be considered as a constant in the integral and so we obtain that: This means that the ensemble average of the product of a property and the number density n is the same as the number density multiplied by the ensemble average of the property alone. Indeed, since n doesn't depend on u, it can be brought out of the integral. In particular, equations (1), (2) and (3) are correct only in case of a strong scale separation, in which the concentration, represented by n, could be considered constant in more realizations; on the contrary in granular flows, this procedure is applicable only to a single realization, as pointed out by Goldhirsh (Goldhirsch 2003). As outlined previously, the central concept is that granular flows are not ergodic systems, as the gases are, for the following reasons:

1. In gases the number density n could be considered constant, since the number of particles tends to infinity and the concentration is not affected significantly by one particle more or less. On the contrary in granular flows, where we have a control volume whose dimensions are comparable to that of particles, one particle more or less may change significantly the value of n.

2. The presence of gradients at the same scale of particles collisions may cause a non-uniform flow along the vertical axis (contrary to the longitudinal axis) so that one realization could be not representative of the whole process, regarding the vertical direction.

Following these arguments, we may say that the kinetic theories average process, based on the ensemble average, could be applied in first approximation to a single realization in granular flows field.

With single realization we refer to a measurement of the flow characteristics made on a time period small enough, so that the hypothesis of the kinetic theories can be considered applicable. In a single realization, the number density n is assumed to be constant and the two types of average still coincide with each other.

3 AVERAGING PROCESSES

From all the above considerations, it becomes clear the fundamental role of the averaging process. The approach adopted by the kinetic theories is statistical and a different way to compute the mean values of the variables may lead to important changes in the final system of equations. In this section, we analyze the types of average that could be applied, trying to indi cate the most appropriate for a granular flow composed by relatively large particles.

3.1 Phasic average and mass-weighted average in a single realization

In the literature two main types of average are men tioned as useful for the two-phase mechanics (Drew 1983): the phasic average and the mass-weighted average, also called Favre average (Favre 1965).

The first is defined as follows:

where the brackets < > indicates a general averaging process (over time, space or ensemble), β indicates the phase and ψ is a general variable of the process.

X (β) is the phase function, which is defined as:

The second type of average, called mass-weighted

average, looks as follows: In this case, as suggested by the definition, the average of the variable ψ is weighted according to the density ρ (β) of the phase. If we now consider the generic average <> to be the ensemble average, done over the whole statistical ensemble of particles of a single realization as in the kinetic theories, we obtain two different results according to the methods previously mentioned. Being N p the number of particles of the realization, by applying the phasic average, we obtain the following form: where i is the i-th particle of the realization. In particular, if we consider the velocity u and the concentration c of the solid phase, we obtain: Equation (8) is the phasic-ensemble average of the velocity and equation (9) is the phasic-ensemble average of the concentration. Regarding the Favre or mass-weighted average, and still considering the ensemble average, we may write it as follows: where i the i-th particle and ρ represents the density of the granular phase, which, in term of concentration, is ρ = c ρ s with ρ s the density of the particles, which is assumed constant. By considering again the velocity and the concentration, we obtain the following relation for the mass-averaged velocity: where, obviously, $\tilde{c} = \hat{c}$. For homogeneous fluid (like pure water), the final averaged values coincide; in the kinetic theories, the assumption of very small size of the particles and a constant concentration (number density n) make the two averages still coincident. However, for granular flows, these two assumptions are no more valid and the two averages may lead to different results. 3.2 Averaging process in case of multiple realizations In cases in which the process is characterized by more than one realization, we need a further averaging procedure. Let us assume, by now, a phasic-average to

define the average of a property in more than one

realization:

where ψ is the phasic-average of a general property

over all the realizations, $\tilde{\psi}$ (k) is the phasic-ensemble

average of the single k-th realization and R is the

number of realizations on which we average.

We want to underline that in making these averages,

we assume that the fluctuations with respect to the

mean values of the single realization are governed by

collisional mechanisms, related to the kinetic theories.

On the other hand, in the case of several realizations,

the fluctuations are related to macroscopic scales too

(like the flow depth or the mean velocity of the flow).

4 EXTENSION OF THE CONTINUITY EQUATION

In the usual application of kinetic theories to gran ular flows, there is no distinction between a single realization and more than one realization: the ensem ble average is applied assuming the same Boltzmann hypothesis of ergodicity of the system.

Starting from the above considerations, we will take into account the influence of the different averaging process, introduced in the previous sections, into the mass, momentum and energy conservation equations.

However, by now, we want to focus our attention on the mass conservation equation, since it is rather simple, and allows us to check the validity of our assumptions with laboratory test on a granular flow in uniform condition.

The mass conservation equation for a dry granu lar flow, that is derivable from the kinetic theories is written as in the following (Jenkins & Savage 1983, Jenkins & Richman 1985):

where n is the number density, m the mass of the single particle, i = 1, 2, 3 indicates the three component of the velocity.

Notice that we can write the equality ρ g = m g c = nm g , where ρ g is the density referred to the granu lar phase, m_g is the mass of the granular particles and c

is the particles concentration. In the following we use

$\rho_g = c \rho_s$ in the place of nm, to be coherent with the

fluid mechanics notation. Then, since m_g is a constant

value, simplifying equation (13) we can rewrite:

Considering a 2D statistically stationary and uniform

channel flow in the longitudinal direction x (Figure 1),

in which, however, we assume that the condition of homogeneity and stationarity is reached in a time interval of the same order of the time scale that governs the kinetic theories (that is, in a single realization), equation (13) reduces to: where y indicates the normal direction. This implies that $< u_y >$ is equal to zero along the entire depth; that is, no vertical component of the mean motion is present. In the case of granular material, since the system is not ergodic, the average of a single realization is not the same of that of more realizations and the average procedure introduced by us must be applied. Consequently, Eq. (13) must be further averaged on all the realizations, that is: Afterwards, we express the variables averaged on a single realization $\tilde{u}^{(k)}_y$ and $\tilde{c}^{(k)}$ as the sum of a mean value and of a fluctuating portion, recalling the Reynolds decomposition: where, by definition $u'_y = c' = 0$, and u_y and c are values averaged according to equation (12). Inserting these terms in (16), we obtain the following equation: Equation (18) in the y-direction implies that $u_y \neq 0$; then it follows that: It is easy to prove that if we had adopted the mass average definition in the averaging processes (eq. 11), instead of eq. (18), we would have obtained the following equation: from which, in this case, results that the mass averaged vertical component of the velocity is $\hat{u}_y = 0$ along the intere flow depth. 4.1 Modeling of the diffusive term This diffusive term $c' u'_y$ will be modelled through a Boussinesq diffusive model, that is:

where is a suitable mass diffusion coefficient.

According to the considerations made in section 2.1,

our aim is to define this diffusive coefficient by mak

ing it proportional to a velocity scale and a length scale.

Regarding the velocity scale, it could be the root mean square of the granular temperature or the gradient of the velocity $\partial u/\partial y$. In fact, these two parameters coincide only if in the kinetic energy balance equation the diffusion process is negligible, that is the produc tion of energy is exactly balanced by the dissipation of energy. However this is not always an acceptable hypothesis (Armanini et al. 2014). A similar problem of scale separation is present even in the kinetic energy budget, in which the same geometrical scale governs the production process and the dissipation process of kinetic energy (Armanini 2013).

We need to verify which scale of the two is representative of our intermediate scale.

Finally, the length scale will probably be the dis tance of the particles from the walls.

In this way, we will able to extend to the intermedi ate scale the closure relation now adopted for the small scale, by simply introduce a different length scale (the distance from the walls instead of the particles diameter).

The coefficient will be independent from the aver age process and will be correctly related to the larger scale fluctuations (larger with respect to the collisional scale).

5 EXPERIMENTAL ANALYSIS AND RESULTS

The experimental investigation is fundamental to ver ify and prove our theoretical hypothesis.

In the Hydraulic Laboratory of the University of Trento, a glass-walled open channel of 3 m length and variable width, from 5 cm to 15 cm, with an adjustable slope has been used to perform several experiments; through a weir in the downstream end of the channel, a deposition layer is created (static bed). The mate rial flowing in the channel was a dry uniform granular material made up of artificial zeolites of almost spher ical shape, with a material density $\rho_s = 1050\ Kgm^{-3}$. The diameter of the particles has been chosen to be between 0.5 and 0.6 mm. These particles flow over the static bed, so that mobile bed condition takes place; in this condition, with a constant discharge in time, we have a stationary regime (Armanini et al. 2003), and the free surface slope and mobile bed slope coincide. Videos of the flow have been acquired through two cameras with a video rate of 2000 or 3000 fps, one at the sidewall and one above the flow free surface. The velocity of the particles and their concentration were derived through the Voronoi method (Spinewine et al. 2003).

Figure 2 shows the behaviour of the measured value of the phasic-averaged normal component of the veloc

ity u y . Except for a thin layer near the free surface, this

velocity component is always negative, indicating a

general motion of the particles downward to the bed. Figure 1. Glass-walled open channel in mobile bed condition. Figure 2. Vertical velocity recorded at the sidewall of the channel. Figure 3. Transversal velocity recorded at the top of the flow. This seems to be consistent with our theoretical considerations. Furthermore, we have analysed the transversal velocity component u z (phasic-ensemble average on more realizations) on the free surface, in order to evaluate the presence of possible secondary circulations and we compared it with the normal component. We noticed that the two motions are not correlated: in fact, as shown in Figure 3, in which z indicates the transversal coordinate, the velocity is negative on the side corresponding to the measurements of Figure 2 (z-coordinate equal to 5). This represents a motion of the particles from the wall to the center of the flow and it is coherent with a presence of a possible secondary circulation that produces, on the lateral wall, an upward flow (positive values of the velocities) and that becomes important on the small layer near to the free surface. Thus, we can conclude that the downward motion observed on the sidewall, is reasonably not due to secondary circulations.

The experimental part is still in progress. Further

experiments with 15 cm of width must be carried out,

in order to compare the motion of the particles along

the side walls and on the top of the flow, with a

ratio of the width over the height L/H >> 1. In this

way, we want to make clear the independence of the

vertical velocity gradient with respect to secondary

circulations.

The experiments will be useful to define what is

a single realization: we want to identify, in our sta tistically stationary process, a single realization as a record long enough to catch a whole collisional event, but short enough with respect to the fluctua tions linked to the spatial gradient. By analyzing the video-registrations, we will compute statistical analy sis to define the time scale corresponding to a single realization.

Finally, we will verify through the experiments the real length scale to which the diffusive coefficient is proportional to. The vertical velocity gradient mea surements will be reproduced by our diffusive model, observing if the distance from the wall is a correct length scale (in alternative the flow depth could be adopted) and what is the most appropriate velocity scale.

6 CONCLUSIONS

Starting from the analysis of the kinetic theories, we observed that in the application to granular flows they still have some weaknesses.

In particular, granular flows lack a strong scale separation, which is a fundamental requirement for the statistical approach adopted in the derivation of the Boltzmann equation and the ensemble average.The number density n is by its own definition constant with respect to the ensemble average; this can be considered true in gases, where n tends to infinity and a few particles do not change it significantly. On the contrary, in granular flows this do not hold anymore and the fluctuations of its values must be taken into account: for this purpose a different scale, that we named intermediate scale, should be considered.

We have shown different types of average, focusing our attention on their influence on the final equations, when dealing with a two-phase flow with particles of dimensions comparable to the dimension of the control volume. In particular, the phasic average and the mass weighted average have been recalled.

In this paper, we propose a new statistical approach to deal with granular flows, adopting two averaging processes, according with the number of realizations: the phasic average is applied firstly to the single realization and then to the whole process, when treating all the recorded realizations. Applying this procedure, a

Sediment transport in the Schelde-estuary: A comparison between measurements, transport formula and numerical models

Y. Plancke

Flanders Hydraulics Research, Antwerp, Belgium

Antwerp University, Antwerp, Belgium

G. Vos

Antea Group, Antwerp, Belgium

Flanders Hydraulics Research, Antwerp, Belgium

ABSTRACT: The Schelde-estuary serves different estuarine functions and therefore faces managers with

multiple challenges: increasing tidal propagation vs. safety against flooding; sedimentation in the navigation

channel vs. port accessibility; changing dynamics vs. ecology. Within the Flemish-Dutch Long Term Vision for

the Schelde-estuary, a 4 year (2014–2017) research programme was defined, in which 8 topics will be dealt with

(e.g. tidal penetration, risk for regime shift, sediment strategies, valuing ecology). Two fundamental tools are

crucial in answering the different questions towards the future management of the estuary: expertise/system

understanding and numerical models. Where the numerical models reproduce the hydrodynamics reasonably

well, sediment transport and the resulting morphological changes is still a big challenge. Therefore an extensive

monitoring campaign was performed in 2014, during which both hydrodynamic and sediment transport measure

ments were performed in the Schelde-estuary.The data allows to validate the existing numerical models, allowing

a better assessment of the possibilities and limitations of the present numerical models. This paper describes the

validation of a 2D numerical model, that is used to optimize the relocation strategy of non-cohesive sediments

in the Beneden-Zeeschelde. The comparison of sediment transport of field data and numerical model results

show a rather promising agreement (i.e. differences of factor 2 to 3) for the Beneden-Zeeschelde. However,

important differences (both in patterns, intensities and between different formula) were found when comparing

topo-bathymetric changes predicted by the morphological

numerical model and observed bathymetric changes.

1 INTRODUCTION AND BACKGROUND

In order to supply managers with adequate answers, research tools (both expertise/system understanding and numerical models (Peters et al., 2006)) are crucial in answering the different questions. Where the numer ical models reproduce the hydrodynamics reasonably well, sediment transport and the resulting morpholog ical changes is still a big challenge. These limitations translate into uncertainties of the results, and trig ger the precautionary principle within environmental impact and appropriate assessment (within the scope of the European Bird and Habitat Directive).

In order to collect appropriate datasets for validation of numerical models and to improve our system under standing, good sediment transport measurements are vital. Nevertheless, measuring sediment transport remains one of the most challenging aspects in river engineering (Plancke et al., 2012a; Thant et al., 2016).

Within the scope of a common Belgian-Dutch research programme ("Agenda for the future"), which deals with system management challenges for the near future (e.g. tidal penetration, risk for regime shift, sediment strategies, valuing ecology), additional measurements were performed in order to validate state-of-the-art numerical models. This paper describes the measurements results, a comparison with classic sediment transport formula and the validation of the numerical models. 2 THE SCHELDE-ESTUARY The Schelde-estuary is a macro-tidal estuary with a length of 180 km in Flanders and the southern part of the Netherlands (Figure 1). The Vlakte van de Raan (“mondingsgebied”) connects the estuary with the North Sea and should be seen as an integral part of the estuary. This part is a shallow water area with several channels. The Vlakte van de Raan (–20 KM to 0 KM) connects to the Westerschelde (KM 0 to KM 60), which has a multiple channel system, with ebb and flood channels and intertidal sandbars in between. More up-estuary, near the Dutch-Belgian border, the morphological system changes into a single channel system, the Zeeschelde (KM 60 to KM 160). The estuary is characterized by semi-diurnal tides, causing ebb and flood currents with important sediment transports of both cohesive and non-cohesive

Figure 1. Overview of the Schelde-estuary and different

locations where field measurements were performed.

sediments. The Schelde-estuary serves different estu

arine functions and therefore faces managers with

multiple challenges: increasing tidal propagation vs.

safety against flooding; sedimentation in the naviga

tion channel vs. port accessibility; changing dynamics

vs. ecology.

To guarantee port accessibility, dredging and relo

cation works are performed on a daily basis. In

order to perform these works, licenses are granted

by the responsible governments (Dutch governments

in the Westerschelde, Flemish government in the

Zeeschelde). Dredged sediment are relocated within

the estuary, in order to minimize possible effects.

Within the Westerschelde mostly sand is dredged,

while in the Beneden-Zeeschelde (near the Antwerp port area) both sand and silt/mud are dredged during maintenance works.

Although annual discharge and sediment transport measurements (going from continuous SPM mea surements at several points to yearly sailed transect measurements) are performed within the scope of the MONEOS programme (Plancke et al., 2012b), additional measurements were necessary for model validation.

3 METHODOLOGY

3.1 Field measurements

In 2014, several measurement campaigns were per formed, during which both hydrodynamic and sedi ment transport measurements were performed in the Schelde-estuary. At more than 10 locations (Figure 1) measurements were executed over a full tidal cycle (13h): 3 locations at the Vlakte van de Raan, 7 loca tions at the Zeeschelde and 1 location at the Rupel, a tributary of the Zeeschelde.

At each location one or two vessels were used. The first vessel was anchored during the measure ment period, performing measurements at a fixed point in the estuary. Currents were measured using ADCP (vertical profile) and Aanderaa Current Meter (point), while sediment transport was measured using both direct (Delft Bottle and pump samples) and indirect (OBS,ABS) techniques.The Delft Bottle technique was used both near-bed (using frame) and in the water column (suspended), at four different positions (bed + 20 cm, bed + 40 cm, bed + 100 cm and bed + 200 cm). Measurements were executed continuously, with sampling times varying from 3 minutes (at peak transport) to 15 minutes (near slack moments). From these measurements total transports were derived every 30 minutes. At those locations where a second vessel was available, additional transects were sailed using ADCP (current and sediment transport fromADCP-backscatter). For the locations at the Vlakte van de Raan, additional frames were placed at the bed, allowing long term (4 weeks) measurements of hydrodynamics (currents and waves) and sediment transport. The measurements are used to validate the available numerical models.A first project deals with large scale sediment management issues in the down-estuarine part, Vlakte van de Raan (Van der Werf et al., 2015). A second project focusses on management strategies for the most up-estuarine part, Boven-Zeeschelde (Vanlede et al., 2015). A third project will detail the future sediment strategy in the Beneden-Zeeschelde and will be discussed in this paper. Both Delft3D and TELEMAC models are used in these different studies. 3.2 Numerical modelling In order to receive a new license to relocate dredged sediments in the Beneden-Zeeschelde, an appropriate assessment is necessary. Within this assessment the possible effects of the relocation strategy must be evaluated, with regard to the reference situation (present strategy). A preliminary study was performed by Flanders Hydraulics Research to optimize the relocation strategy, both for sand and silt/mud, taking into account present and future challenges (tidal penetration, possibility of a regime shift) and minimizing possible effects on primary production, birds and fishes, etc. A 2D numerical model of the Beneden-Zeeschelde was set-up in Delft3D, based on existing state-of-theart NeVla-model (Figure 2). Initially, it was opted to make a distinction between the cohesive (silt/mud) and non-cohesive (sand) sediment transport model, which is state-of-the-art (Dam et al., 2013). Possible effects are different: relocation of silt/mud will have a possible effect on the suspended sediment concentration and therefore will influence the primary production; relocation of sand will have an effect on the morphology and therefore will influence the ecological valuable habitats. However, as both sand and silt transport take place simultaneous in reality, the final model included both the cohesive and the non-cohesive sediment. It was found that mainly the cohesive sediment transport was influenced by this approach, as the major part of the available transport capacity was consumed by the non-cohesive sediment transport. The numerical model has a grid resolution in the zone of interest of 25 m (L) by 15 m (W) (width of

Figure 2. Overview of model grid: existing NeVla grid (green) and detail (3 × 3 refined) grid for this study (red).

the estuary in this zone ~500 m). It is driven by a water level time-series at the down-estuarine boundary, while a discharge time-series (generated from the NeVla-model) was defined at the up-estuarine boundary. A uniform Manning bed roughness was applied (0.025 $m^{1/3}/s$), resulting in a good agreement of water levels in the study area.

A sensitivity analysis was performed on the sediment transport model (both cohesive as non-cohesive) and the morphological model. As the additional measurements focused on non-cohesive transport, only this part is discussed in the paper.

3.3 Sediment transport formula

Several formula are available to describe the sediment transport in rivers and estuaries. Within this study it was chosen to use both Engelund-Hansen formula (1) and Van Rijn formula (2a and 2b)), which are both suited for non-cohesive sediment transport of fine sand. Further information on these formula can be found in (Engelund & Hansen, 1967) and (Van Rijn, 1993).

It can be seen that concept of these formula is differ

ent (total load q_t vs. bed q_b and suspended q_s load) and

the different effect of both mean flow velocity u (power

5 vs. power 3) and median grain size d_{50} (inverse vs.

proportional) between these formula.

4 RESULTS

4.1 Field measurements

Figure 3 shows the hydrodynamic conditions for the

different locations in the Zeeschelde. The tidal char

acteristics change significantly along the estuary:

the most down-estuarine locations of the Zeeschelde Figure 3. Overview of hydrodynamic conditions: water level (top) and flow velocities (bottom) in the Zeeschelde. Figure 4. Overview of sediment transport in the Zeeschelde. (Liefkenshoek, Oosterweel) have an almost symmetrical tide; more up-estuary the tidal asymmetry increases, most pronounced at Schellebelle and Schoonaarde. The tidal range increases from the North Sea up to Driegoten due to the funnel shape of the estuary. More up-estuary it decreases due to the damping effect of the undeeper channels. The asymmetry in the vertical tide is translated into the horizontal tide (flow velocities). Highest flow velocities are found at Oosterweel (ebb and flood phase) and Driegoten (flood phase). Figure 4 shows the sediment transport for the different locations in the Zeeschelde. The sediment D50 ranges from very fine sand (near-bed “DBF”) to very fine silt or mud (suspended “SUSP”). Sediment transport patterns are different for different locations, which is related to the position along the

estuary and its position on the transect. For Driegoten

the sediment transport shows a maximum 1 to 3 hours

after low water. The sediment during this period is

muddy, with a lot of flocs. Where the flow velocities in this period are low, the peak is probably related to the technique of the Delft Bottle: while this technique is suited for sand, it was found that during period with low flow velocities, currents through the bottle are insuf ficient to transport mud through the bottle, leading to anomalies in the measured transport.

4.2 Numerical model

The numerical model was validated with the available field measurements. Where the simulated period was not identical to the moment of the field measurements (due to limited availability of boundary conditions of fresh water discharge), a tidal cycle with simi lar characteristics was searched during the simulation period. A good agreement was found, although small differences are possible.

Figure 5 (top panel) compares the measured and modelled flow velocities at Kruibeke. It can be seen that during ebb (HW-360’ to HW + 30’) good agree ment is found. During flood, peak velocities are overes timated in the model.A possible explanation was found in (1) a steeper rising of the tide in the model and (2) a difference in position along the cross-section due to grid size resolution (measurement was performed out side the navigation channel and thus relatively close to the bank (steeper bed slope)).

Figure 5 (bottom panel) compares the measured and modelled sediment transport rates at Kruibeke. It should be mentioned that sediment transport mea surements were executed at several (4) points over the vertical profile, and an assumption (profile derived from relative profiles of all measurements divided in 30'-blocks of the tidal cycle) had to be made to inte grate this towards a transport over the full vertical. It can been seen that both the Engelund-Hansen and the Van Rijn formula give similar patterns over time, with higher values from the Van Rijn formula at peak transport. When model results are compared with the measurements, some similarities can be seen, although difference are present: At the start of the ebb phase (HW-300') a peak is found in the measurements, which does not occur in the model. However, it is not certain that this peak is related to natural transport (an impor tant increase in flow velocity coincides with this peak), or other factors (e.g. re-suspension due to ships) play a role. During the next phase of the ebb, a similar pat tern is found, with modelled transport rates being twice as high as measured values. During the first phase of the flood period (until HW + 200'), modelled trans port rates are lower than measured values, and at peak transport differences are even larger. This is proba bly related to the difference in flow velocity at this moment, as sediment transport rates relate to the 5th power (Engelund-Hansen) of the velocity. Although differences exist, it was concluded that the model gave

Figure 5. Comparison between measurements and numerical model of flow velocity (top) and sediment transport (bottom) at Kruibeke. Figure 6. Difference (left) and consistency (right) map of bathymetry of Beneden-Zeeschelde over period 2009–2014 [red = erosion, green = sedimentation]. a reasonable agreement with regard to non-cohesive sediment transport. In the next phase, morphological updates were activated in the model. To evaluate the model, several topo-bathymetric maps (interval of one year) of the Zeeschelde were used and both difference and consistency (to identify yearly variation in sedimentationerosion patterns) maps (Figure 6) were derived. From these maps, it can be seen that several zones in the Beneden-Zeeschelde are eroding (red), while other zones are characterized by sedimentation (green).

Figure 7. Difference map (bathymetry) from numerical model using Van Rijn (left) and Engelund-Hansen (right) formula [red = erosion, green = sedimentation].

Morphological changes were computed by the numer ical models (Figure 7), using both Engelund-Hansen and Van Rijn formula. It can be seen that (1) patterns don’t show a good agreement, (2) that the intensity of bathymetric changes is larger in the model than in reality and (3) different formula give very important differences in erosion-sedimentation patterns. It can be seen that the Van Rijn formula predict erosion in the channels and accretion near the banks, while the Engelund-Hansen formula predicts the opposite.

Based on these results, it was decided that sce narios within the project to optimize the relocation strategy could not be done using the morphological model. Therefore different scenarios were investigated by making changes in the initial bathymetry (adding sediment to reproduce the relocation) and analyzing changes in flow patterns and sediment transport rates.

5 CONCLUSIONS

In 2014, a series of field measurement campaigns were organized to collect datasets for numerical model calibration and validation with regard to sediment transport and morphology. In the past, the models have been extensively calibrated and validated for hydrody namics, but due to lack of available measurement data, sediment transport was never really validated.As mea suring sediment transport remains very challenging, results have an uncertainty due to (a) measurements techniques (e.g. sensitivity of indirect techniques to sediment properties, bio-fouling), (b) measurement execution (e.g. errors made during direct sampling in sample collection), (c) field conditions (external factors e.g. ships) and (d) data post-processing (e.g. calculation of vertical profile based on discrete data). Within the scope of several projects, these new datasets were used to validate numerical models. Within this paper the results of a validation was

described for a 2D numerical model in Delft3D

used to optimize the relocation strategy of dredged

Thant, S.; Plancke, Y.; Claeys, S. (2016). Sediment transport

measurements in the Schelde-estuary: how do acous

tic backscatter, optical transmission and direct sampling

compare? (proposed for the 4th IAHR Europe Congress).

Vanlede, J.; Smolders, S.; Maximova, T.; Teles, M.J. (2015).

The unstructured Scaldis model: a new 3D high resolu

tion model for hydrodynamics and sediment transport in

the tidal Scheldt, in: (2015). Scheldt Estuary: physics and

integrated management Special Session on of the 36th

IAHR WORLD CONGRESS, 28 June–3 July, 2015, The

Hague, the Netherlands. van der Werf, J.; van Oyen, T.; De Maerschalck, B.; Nnafie, A.; van Rooijen, A.; Taal, M.; Tonnon, P.K.; Verwaest, T.; deVet, L.;Vroom, J.;Van der Wegen, M. (2015). Modeling the hydro-morphodynamics of the mouth of the Scheldt estuary Special Session on of the 36th IAHR WORLD CONGRESS, 28 June – 3 July, 2015, The Hague, the Netherlands. Van Rijn, L.C. (1993). Principles of sediment transport in rivers, estuaries and coastal seas. Aqua Publication, The Netherlands, Amsterdam. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Influence of non-uniform flow conditions on riverbed stability:

The case of smooth-to-rough transitions

D. Duma, S. Erpicum, P. Archambeau, M. Pirotton & B. Dewals

Research Group Hydraulics in Environmental and Civil Engineering (HECE), ArGEnCo Department,

University of Liège (ULg), Liège, Belgium

ABSTRACT

Predicting the entrainment of bed material by water flow is of utmost practical relevance, particularly for the safe design of hydraulic structures and to ensure riverbank stability. The inception of sediment motion was studied extensively for a broad range of bed material properties and bed slopes; but most of these previous studies were based on the assumption of uni form flow conditions. In particular, many of them used a friction formula for evaluating the bed shear stress. Recently, Hoan et al. (2011) and a few other researchers explored new avenues to assess the sta bility of riverbed material. They linked directly the sediment pick-up rate to the local flow and turbu lence characteristics. Such approaches avoid the need for closure relations valid only under uniform flow conditions, such as most friction formula. However, they were validated only for a limited range of flow conditions and geometric setups (e.g., for sudden expansions).

In the present research, we aim to investigate whether such a new approach also holds in the case of a different geometric setup, namely for the case of smooth-to-rough transitions (Fig. 1). This configura tion is very often observed downstream of hydraulic

structures, at the transition between the concrete lined bottom and the natural riverbed.

We performed flume experiments considering two different configurations. We used a uniform bed mate rial to define a reference configuration (C1) and we compared to a sudden transition from smooth to rough bottom (C2, Fig. 1). Two different flumes were used to assess possible scale effects and to extend the range of tested flow conditions. The bed material (diam eter d = 8, 15 and 30 mm, relative density �= 0.5, 0.7 and 1.65), the flume slope and the flow rate were varied. Using a Vectrino II ADVP (manufac tured by Nortek) and a UVP probe (manufactured by METFLOW), flow velocity was measured at a high frequency (100 Hz) along different profiles in the near field of the smooth-to-rough transition. The sediment pick-up rate was evaluated by counting the number of displaced particles.

Measurements of flow velocity and turbulent fluc

tuations were used to compute the non-dimensional Figure 1. Tested geometric configuration for smooth-torough transition (side view). Figure 2. Observed bed mobility parameter E , as a function of Shields parameter ! S1b (left) and the bed stability parameter ! u−σ (u) proposed by Hoan et al. (2011, right). stability parameters ! u−σ (u) introduced by Hoan et al. (2011). The sediment pick-up rate was expressed in non-dimensional form through the bed mobility parameter E , also defined by Hoan et al. (2011). In Fig. 2a (respect. Fig. 2b), we display the observed bed mobility parameter as a function of the standard Shields parameter ! S1b (respect. the bed stability parameter ! u-σ (u) of Hoan et al., 2011). Despite some scatter in the experimental results, they suggest a more significant correlation between the mobility parameter E and the new bed stability parameter ! u-σ (u) ; rather than between E and Shields parameter ! S1b , which was evaluated assuming uniform flow conditions. This study confirms the potential of this new approach based on local flow and turbulence characteristics. Additional experiments remain necessary to parametrize the relationships between the bed mobility parameter E and bed stability parameters, which were tested so far only in a narrow range of setups. REFERENCE Hoan, N. T., Stive, M., Booij, R., Hofland, B., Verhagen, H. J. (2011). Stone Stability in Nonuniform Flow. Journal of Hydraulic Engineering. 137(9): 884–893. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

The probabilistic solution of dike breaching due to overtopping

Z. Alhasan, D. Duchan & J. Rˇîha

Institute of Water Structures, Brno University of Technology, Czech Republic

ABSTRACT: This paper contains the results of the probabilistic solution of the breaching of a left bank dike

of the River Dyje at a location adjacent to the village of Ladna near the town of Breclav in the Czech Republic.

A mathematical model describing the overtopping and breaching processes was proposed so as to obtain the

solution. With this model, the overtopping of a dike is simulated using simple surface hydraulics equations.

Dike breaching commences with the exceedance of the erosion resistance of the dike surface. For the modelling

of dike erosion, a simple transport equation was used with erosion parameters calibrated using data from past

real embankment failures. The sensitivity analysis of involved uncertain parameters was carried out using the

screening method. In order to achieve the probabilistic solution, the Latin Hypercube Sampling method was

used to generate plausible sets of random values for the sensitive variables. These random values were derived from a multidimensional distribution. Typical phases of the dike breaching process due to overtopping were distinguished. The final results of this study take the form of probabilities for those typical dike breach phases.

1 INTRODUCTION

Dikes on riversides protect people and property against the destructive effects of floods. Potentially endan gered areas cannot be absolutely protected against floods due to the fact that no dike design has 100% reliability. Dike overtopping is the most frequent cause of dike failure. According to statistics regard ing embankment dam and dike failures, this kind of failure represents about 40% of all embankment dam and dike failures (Jandora & R˘îha 2008).

The outputs of dike failure process modelling are used in the drafting of flood control plans containing predictions of the extent of potential floods and their parameters, such as the water level and the progres sion of the flood over time in the area behind the dike.

The mechanism by which embankment dam failures occur due to overtopping was analysed in studies such as (Fread 1988, Singh 1996) and others. Singh (1996) presented types and causes of dam failures, discussing the hydraulics of dam breaching, models with dimen sional and dimensionless solutions. He also performed a comparison of dam breach models.

Wahl (1997) examined empirical procedures and a numerical model for predicting breach parame ters, and outlined a program for improving numer ical models for the simulation of embankment dam breaching. Jun & Oh (1998) presented a sim ulation of earthen dam-break processes due to overtopping, and performed flood routing analy sis downstream of the breach using the NWS DAMBRK code. Coleman et al. (2002) presented breaching experiments on homogeneous embank ments made of non-cohesive materials breached under a constant reservoir level. Chinnarasri et al. (2003) investigated the flow patterns and progressive damage during dike overtopping after analysing the data obtained from nine experimental runs. An experimental study of embankment dam breaching was performed by Dupont et al. (2007), whose laboratory tests enabled the validation of a numerical model. An overview of field tests and laboratory experiments was given by Morris et al. (2007) under the IMPACT project, which addressed the assessment of risks from extreme flooding caused by natural events or the failure of dams. Series of tests involving dike breaches due to overtopping were performed by Schmocker & Hager (2009) to examine model limitations. An experimental model of a dike-break induced flow was designed by Roger et al. (2009), who compared experimental data with the results of numerical computations. One-, twoand three-dimensional dam break models are discussed by Wang & Bowles (2006), who developed a 3D model of non-cohesive earth dam breaching. A oneand two-dimensional numerical dam-break model is provided by Galoie & Zenz (2011), who solved shallow water equations using the Finite Differences Method (FDM). Information about dam failures is necessary for the verification of numerical dam-break models. Lemperiere et al. (2006) presented a database of case studies of embankment dam failures and summarized the lessons learnt from the failures due to overtopping in order to propose a new empirical formula for breach peak outflow. Løvoll (2006) presented the results of 3 field tests carried out on 6 m high embankment dams in Norway during the period of 2001–2003. In the publications mentioned above the reliability of dams or flood protection dikes is assessed by estimating the probability of failure related to the

Figure 1. Time-related parameters.

return period of the design flood. Detailed studies assessing the probability of dike failure and its dam age in case of overtopping are still not available in the literature.Assessment of the probability of total and/or partial dike breaching which is an input into the flood risk analysis of the area behind the dike was the main purpose of this study.

2 TIME RELATED PARAMETERS

The time course of a dike breach from the flood begin ning until the total dike collapse is described by the following parameters (Fig. 1):

- The time of the flood beginning: t = 0.
- The flood wave arrival duration is the period during which the water level increases in the river but does not exceed the dike crest (no overtopping).
- The time t o of the start of overtopping is the instant when water begins to overflow the dike crest.
- The resistance phase lasts from the instant of dike overtopping until dike erosion begins. Here the water flowing over the dike crest does not initiate

erosion of the dike body. This period is attributed to the resistance of the crest and downstream slope.

– The time t e of the beginning of dike erosion is the instant when the water flowing over the dike crest exceeds the resistance of the dike material.

– The duration of the breach initiation begins with the first erosion of the dike body and ends at the beginning of the dike breaching phase. During this phase, backward erosion of the dike initiates while the crest elevation does not decrease.

– The time t b of breach initiation is the instant when the backward erosion of the crest reaches its upstream edge. From this time dike collapse occurs.

– The breach formation phase is the period from the beginning of dike breaching until the end of the flood. During this period the maximum dis charge and breach size are reached. At time t max the maximum breach discharge Q bmax and maximum breach size are attained.

3 RESISTANCE OF DIKE SURFACE

The main reason for a dike breach due to overtopping is

the surface erosion of the dike material. The resistance
Figure 2. Non-scouring velocity for selected surfaces related to overflowing time (Floods and Reservoir Safety 1996). of dike materials against erosion governs the length of the resistance phase. Erosion of soil particles starts when the water flowinduced shear stress or the flow velocity exceed their critical values related to the dike

material or its surface lining. The resistance is usually defined either by the critical shear stress, critical velocity or critical flow rate (Linford & Saunders 1967, Knauss 1979, Clopper & Chen 1987). Several authors have proposed empirical formulae based on laboratory and field measurements to express these critical values. The resistance for various materials against water flow is summarized within (Floods and Reservoir Safety 1996); see Figure 2. In cases when the dike crest and the surface of the downstream slope are protected by riprap or a grass layer, the erosion of the dike material will start after the protective layer is damaged. The results of research regarding the resistance of grass protective layers were also published within (Hanson et al. 1999). In general, the resistance of the dike materials on the downstream slope can be assessed by comparing the following variables with their critical values: – Shear stress (Linford & Saunders 1967), – Flow velocity (Hartung & Scheuerlein 1970), (Floods and Reservoir Safety 1996), – Froude number (Knauss 1979), – Specific discharge (Knauss 1979). The graphs in Figure 2 can be used for assessing the resistance against erosion of particular types of dike

Figure 3. Diagram of the failure of a dike due to overtopping.

surface linings using the limit (non-scouring) cross

sectional velocity. The limit cross-sectional velocity

was defined as a function of the overflow duration

namely in the case of non-rigid materials (grass,

meshes, mats, etc.).

4 CONCEPTUAL APPROACH

The dike breaching process can be analysed and

divided into its sequential events, allowing the follow

ing typical phases to be distinguished (Fig. 3) (Singh

1996), (Jandora & Rˇíha 2008):

1. Flood wave arrival (no overtopping): due to the

flood event, water level in the river gradually increases but does not exceed the dike crest.

2. Resistance (overtopping no erosion): the dike material or the protective layer on the downstream slope resists the overflow for some time. The resis tance mainly depends on the overflow velocity, the dike material and the protective layer situated on the downstream slope.

3. Breach initiation (erosion – no breaching): grad ual breach formation at the downstream slope and dike crest is initiated due to local erosion when the non-scouring velocity of the surface resis tance of the downstream slope is exceeded. In this phase, a small portion of the dike material from the downstream slope and the dike crest will be breached. This phase represents the duration from the beginning of the downstream slope erosion until the upstream slope is reached. In this phase, the breach bottom elevation approximately remains equal to the dike crest elevation.

4. Breach formation (dike collapse): this phase rep resents the breaching of the dike due to backward erosion of the upstream slope. Usually when the backward erosion reaches the upstream slope, a rapid increase in the discharge through the breach initiates, which causes more intensive erosion of

the upstream slope. During this phase, it is notice able that there is a significant lateral widening of the breach opening and that a considerable amount of the dike material is being flushed away. The ele vation of the breach bottom may reach the terrain elevation of the area downstream of the dike, and the lateral widening continues until the end of the flood event. The problem of dike breaching due to overtopping involves hydraulic and erosion transport phenomena. Therefore, in this study the dike breaching was divided into the process of dike overtopping followed by the process of gradual erosion of the dike material. During both overtopping and erosion, the hydraulic and erosion phenomena are complex three-dimensional processes that involve extremely turbulent three-dimensional flow comprising a mixture of water, air and soil, all with different densities. This fact creates theoretical and mathematical difficulties when solving practical problems. Therefore, the following extensive simplifications were taken into account when proposing the mathematical model: – The 3D process of dike breaching is approximated by a 1D model. – The breaching starts at the lowest point of the dike crest where the first overtopping occurs. – The overtopping width along the dike crest is suggested as an initial value (b_0). This value remains constant during the dike overtopping until the erosion starts. – The resistance against surface erosion is evaluated with respect to the velocity of water flowing at the downstream slope. The limit cross-sectional velocity (Fig. 2) is used for this evaluation. – Parallel gradual backward erosion of the downstream slope is assumed, as is shown in the diagram in Figure 3 (Fread 1988). – The shape of the breach opening is approximated by a rectangle. Dike erosion progress is in both the downward and lateral direction. During the erosion, the bottom of the breach opening remains horizontal and the sides remain vertical. – Water flow along the downstream slope is approximated by uniform and quasi-steady flow (Singh 1996), (Jandora & Rˇîha 2008). 5 MODEL OF DIKE BREACHING 5.1 Conceptual model When constituting the mathematical model, the assumptions described in Section 4 were adopted.

The mathematical analysis of the problem of dike breaching due to overtopping involves the determina tion of time-dependent variables and the proposal of a mathematical model for the solution of those vari ables. Since the problem of dike breaching was divided into two processes, the mathematical model consists of two parts: a hydraulic module which describes the hydraulics of water flow during the dike overtopping process, and an erosion module that describes the progress of the erosion of the dike material.

Field and also laboratory measurements show (Jan dora & Rˇîha 2008) that in practical computations the flow along the downstream slope may be assumed to be 1D, quasi-steady and uniform. No submergence from the downstream water level behind the dike was antici pated. Uniform erosion along the breach bed and sides was also assumed.

5.2 Mathematical model

A. The hydraulic module:

To describe water flow during dike overtopping the following state variables have to be determined:

- $Q_b(t)$ = flow discharge over the dike crest, or through the breach opening;
- $b(t)$ = overtopping width before the erosion starts (= b_0) or breach opening width during the erosion process (determined by the erosion module);
- $h(t)$ = overflow head;

– $v_f(t)$ = mean cross-sectional flow velocity at the

downstream slope;

– $h_f(t)$ = water depth along the downstream slope

(Fig. 3).

Overflow head $h(t)$ is determined as the difference

between the water level in the river $h_s(t)$ and the eleva

tion of dike crest Z_c (or the elevation of breach opening

bottom $Z(t)$ during the erosion process):

where h_s = considered as constant along the breach.

Water flow over the crest is given by the equation:

where m = the discharge coefficient (for a broad

crested weir); and g = the acceleration of gravity.

The water depth h_f of the flow on the downstream

slope can be derived from the Chezy formula:

where β = the angle of the downstream slope; and

n = Manning's roughness coefficient. Flow velocity v_f at the downstream slope is given by the Chezy formula: The initial conditions for the overtopping hold: where b_0 determined as the idealized initial width of the dike crest depression at the overtopping location. B. Erosion module: A simple 1D mathematical model was proposed for modelling the erosion process as it affects the dike body. Unknown variables in the erosion model are: – $b(t)$ = the breach opening width; – $Z(t)$ = the elevation of the dike crest or the highest point of the breach opening bottom. After exceeding the dike surface resistance ($v_f > v_{non}$), the elevation $Z(t)$ is determined by Equation 6 using the erosion module. During the dike breaching, two major processes are distinguished: downward erosion of the breach bottom due to the bed shear stress and breach widening due to lateral erosion and landslides that may occur at both sides of the breach. In general, the processes are three-dimensional and time dependent. To simplify complicated turbulent flow with pulsations and vortices accompanied with landslides of the breach banks, simple equation justified by Singh and Scarlatros (1988), Singh (1996) was used to calculate the change in the breach bottom elevation: This equation is formally consistent with Du Boys' bedload formula (Singh 1996). However, erosion rate depends also on other factors than velocity and can be formulated differently. Similar equation substituting both lateral erosion and landslides of the breach slopes was used for the change in breach width (Jandora & Rˇíha 2008): where dZ/dt = the instantaneous change in the elevation of the breach opening bottom; db/dt = the instantaneous change in breach opening width; t = the time when $t > t_b$ (Fig. 1); and α_1 and α_2 = empirical coefficients expressing the erodibility of the dike material. The value of α_1 can be determined by analysing real dam failure records and the value of α_2 can be estimated within the interval $\langle \alpha_1/20; \alpha_1/5 \rangle$. The range of the values of α_2 was derived from the backward analysis using final breach dimensions of real dikes

failed during the flood incidents in years 1997, 2002

and 2006 in the Czech Republic.

The initial conditions hold:

where Z_c = the elevation of the dike crest.

5.3 Schematization of the flood wave

The water level in the river $h_s(t)$ was derived from

the flood wave passing through the studied profile. To

determine $h_s(t)$ it is necessary to know the character

istics of the flood wave, and the channel rating curve.

The flood wave is generally characterized by its peak

discharge, volume and shape.

Because of the different morphology of individual

catchments and the variability of climatic conditions,

the shape of the flood hydrograph Q(t) is difficult to

generalize. The typical shape of the flood wave is also affected by seasonal periods, snow melting, etc.

In this study the flood hydrograph was approximated by a trapezoidal shape (Fig. 4). This approach is sufficiently variable to be able to describe the hydrographs of various flood waves. The approximated shape of the flood wave was schematically represented by the peak discharge Q_N and by three sections as follows (Fig. 4):

1. The ascending limb reflects the increase in the discharge due to the flood wave's arrival. A linear increase in the discharge over time was assumed. Time interval t_k starts from the instant of the flood wave's arrival and lasts until the instant when the peak discharge Q_N is reached. It varies for each individual flood event, stream and catchment.

2. The horizontal limb approximates the duration of the peak discharge Q_N. Time interval t_d specifies the duration from when the peak discharge is reached until the beginning of the falling limb. t_d can last less than one hour or may exceed several hours or days.

3. The descending limb represents the gradual decrease in the flood discharge. A linear decrease in the discharge over time was assumed.

The value of the peak discharge Q_N of the flood wave is provided by the Czech Hydro-meteorological

Institute (CHMI) for an N -year flood frequency for a

given river profile in the Czech Republic.

6 PROBABILITY OF DIKE BREACHING DUE TO OVERTOPPING

6.1 Uncertainty in input parameters

The uncertainty in the input parameters of the dike

breaching model was taken into account in order to Figure 4. Schematization of the flood hydrograph. obtain a probabilistic solution for the problem. The relevant parameters were classified into three groups: 1. Parameters describing the flood wave: Q_N , t_k , t_d (Fig. 4). 2. Parameters of the hydraulic module: b_0 , m, n (Equations 5, 2, 3). 3. Parameters of the erosion module: v_{non} , α_1 , α_2 (Equations 6, 7). 6.2 Sensitivity analysis The influence of the input parameters mentioned above on the outputs of the dike breaching problem was taken into consideration when selecting parameters for random sampling. For this purpose, the screening method was used to identify the non-influential input parameters (Iooss & Lemaitre 2014). Due to the limited paper extent the sensitivity analysis is not discussed in this paper in more details and only results are mentioned in Section 7. 6.3 Estimation of the probability of dike breaching The assessment of the probability was related to the typical phases of the dike breaching specified in Section 4 (Fig. 3). For the probabilistic solution, a random sampling procedure was used where a set of simulations of dike breaching was generated with the consideration that the value of each uncertain input parameter changes within an interval of values with a specific probability distribution. Using the Latin Hypercube Sampling (LHS) procedure, the sets of input values were randomly sampled and applied in the deterministic model to generate output parameters. The probability P_i of each typical i-th phase of the dike breach was estimated by frequency analysis: 6.4 Description of the algorithm The algorithm describing the procedure of dike breaching consists of the following sub-problems: 1. Definition of the flood wave hydrograph.

Figure 5. Dike’s cross-section at the location of a potential breach.

Table 1. Values of N and Q_N (provided by the CHMI).

N	1	2	5	10	20
Q_N (m^3/s)	160	231	341	436	541
N	50	100	500	1000	10000
Q_N (m^3/s)	693	820	1154	1320	1920

2. Determination of overflow head using a rating curve at the river profile.
3. Breach discharge determined from the overflow (Equation 2).
4. Flow characteristics along the breach approximated by a 1D model of steady uniform flow.
5. Simulation of backward erosion (Fig. 3).
6. Determination of the breach opening size (Equations 6, 7).

7 CASE STUDY

7.1 Description of the studied dike

The studied dike is located on the left bank of the Dyje River at the stationing about 28.8 km. This location is near the town of Breclav in the Czech Republic. A diagram of the dike's cross-section and dimensions is shown in Figure 5. The location of the potential overtopping and subsequent breaching was selected at the lowest point on the dike crest.

7.2 Specification of the flood wave

Return periods N , corresponding flood peak dis charges Q_N and other parameters were set up for the

chosen locality (see Section 7.1):

1. Values of peak discharge Q_N provided by the CHMI

are summarized in Table 1.

2. The duration of t_k was considered to range within

the interval <48; 120> [hour] derived from the

floods in 2002, 2006 and 2006 in the Dyje River.

3. The duration t_d was assumed from 0 to 120 hours

based on data obtained from past flood events. 4. The duration of the descending limb was determined based on typical observed hydrograph shapes of past flood events to be $3.t_k$. 7.3 Sensitivity analysis In the sensitivity analysis the influence of input parameters Q_N , t_k , t_d , b_0 , m, n, v_{non} , α_1 , α_2 on the output variable Q_{bmax} was assessed with the conclusions: 1. Parameters Q_N , t_k , t_d , m, n andα_2 are the most influential. Q_N has the highest influence on the output variable Q_{bmax} . t_k , t_d , m and α_2 have lower influence than Q_N , and n has reverse influence. 2. Parameter v_{non} has only minor influence and parameters b_0 and α_1 have practically no effect on the output variable Q_{bmax} . 3. As a result, parameters b_0 , α_1 and v_{non} may be considered as deterministic. The changes to them have only a minor influence on the resulting Q_{bmax} . The input parameters Q_N , t_k , t_d , m, n and α_2 are the most influential, so their range and their probability distribution should be taken into account during the probabilistic solution of the dike breaching problem. 7.4 Computational algorithm The dike breaching is a dynamic process in which the breach discharge depends on the breach opening size (the breach bottom elevation Z(t) and the breach width b(t)). The breach development depends on the flow velocity. The estimated changes (ΔZ , Δb) are initial inputs for the iteration in each time step.The algorithm consists of the following steps: 1. The probability P of the discharges Q_N was determined using the annual exceedance probability p_N : 2. Defining the flood wave parameters: The peak discharge and flood duration should be determined in order to obtain the flood hydrograph (Q, t). The flood wave parameters are as follows: – Q_N is randomly sampled by the LHS method.

Table 2. Probability values for the typical phases and

comparisons with the values obtained from Equation 10.

	p_N (Equation 10)	No overtopping	Overtopping – no erosion	Erosion – no collapse	Collapse
$Z_c = h_s(Q_{10})$	0.90484	0.90595	0.00378	0.00003	0.09024
$Z_c = h_s(Q_{20})$	0.95123	0.95688	0.00198	0.00002	0.04112
$Z_c = h_s(Q_{50})$	0.98020	0.98543	0.00189	0.00001	0.01267

– t_k is a value randomly chosen using the LHS method from the interval <48; 120> [hours].

– t_d is a value randomly chosen using the LHS method from the interval <0; 120> [hours]. – The parameters t_k, t_d, m, n, α_2 were considered random variables. As there were no reliable data for the analysis of their probability density function, their probability density was set to be uniform.

3. The water level in the stream (the Dyje River) is determined from the instant discharge in the river by the use of the rating curve (h_s, Q).

4. The lowest dike crest elevation Z_c was adjusted to be equal to the stream water level corresponding to the peak discharges with the return periods N = 10, 20 and 50 years, i.e. $Z_c = h_s(Q_{10}, Q_{20}, Q_{50})$.

This enabled the parametric assessment of the probabilities related to the different flood protection levels.

5. Testing whether the stream water level h_s exceeds the crest level Z_c was done using the hydraulic module. The random variables m and n were determined using the LHS, where m values were randomly chosen from the interval <0.3; 0.4> and n val

ues from <0.025; 0.045>. Other parameters were

$b_0 = 2$ [m], $g = 9.81$ [m/s^2] and $\beta = 19.43$ [degree].

6. Testing whether flow velocity at the downstream slope v_f exceeds the non-scouring velocity v_{non}. If $v_f > v_{non}$, calculation of the erosion module was performed, where the instantaneous changes in the breach bottom elevation and breach width were calculated using Equations 6, 7. α_1 was used as a constant ($\alpha_1 = 0.0005$), the α_2 value was randomly chosen from the interval <0.000025; 0.0001>.

7. If the breach bottom elevation reaches the terrain elevation and still $v_f > v_{non}$, only the breach width increases.

8. The procedure described in points 5–7 repeats until the water level in the stream decreases together with the breaching velocity and erosion stops.

9. Calculating the probability of each typical phase of dike breaching due to overtopping was carried out using Equation 9. For the random sampling 5.10^6 simulations were carried out.

8 RESULTS

The final results are presented as probabilities related

to the annual occurrence of a given breaching phase Figure 6. Probabilities [%] of the typical phases of dike breaching due to overtopping. (Section 4). The assessment was carried out for three flood protection levels corresponding to N = 10, 20 and 50 years. The dike was covered by plain grass, which is a standard protection of the slopes. The probabilities are shown in Table 2 and Figure 6. 9 CONCLUSION The main goal of this study was the probability estimation of individual phases of dike breaching due to overtopping. For this purpose, a simple mathematical model of dike overtopping and erosion was proposed, and the resistance of dike cover on the downstream slope and the erosion criteria for dike material were defined. A sensitivity analysis was performed, a deterministic mathematical model was developed and statistical modelling based on the LHS method was carried out in order to calculate the probability of dike breaching. The statistical modelling was carried out for 5 million simulations. The sensitivity analysis results indicated that the Q_N, t_k, t_d, m, n, and α_2 are the most influential parameters and should be taken into account as random variables. The resulting peak breach discharge (Q_{bmax}) was not sensitive to parameters b_0, v_{non} and α_1. This can be attributed to the very long duration of the flood waves simulated in this work, which corresponds to a river profile lying in the lower portion of a catchment. The assessment was performed for a dike covered with plain grass – poor cover, and the dike crest elevation was specified for three different design discharge values in the Dyje River as given in Table 2. The results for the “no overtopping” phase are in good agreement with the “accurate” results obtained from Equation 10. The small values gained for the

probability of the dike erosion – no collapse phase

can be attributed to the very long duration of the flood

waves in the Dyje River simulated in this work.

During the study, numerous specific practical and

theoretical problems were solved. These will be solved

in more detail during further research:

– Comprehensive sensitivity analysis including more

output variables should be carried out to study the

influence of erodibility parameters on the breaching

process in more detail.

– Due to the large number of simulations, the com puting time needed for this study was extensive, usually exceeding 4 days for one set of simulations. It is necessary to search for more efficient sampling methods. This will open up the possibility of using more complex dike-breach simulation techniques, including e.g. 2D models.

– An initiative to compile a database of dike fail ures, dike materials, resistances of individual lining materials, etc. should provide information for the development of more reliable probability distribu tions of individual input random variables.

This study indicates the potential of the probabilistic assessment of dike failures. In practical cases it can help to indicate the most vulnerable reaches of dikes and to propose improvements to be made to these dikes at such reaches by installing more resistant linings or designing emergency spillways for the dikes.

ACKNOWLEDGEMENT

This paper has been prepared under the projects TA04020670 The application of reliability methods in dam safety supervision of hydro-engineering struc tures in relation to their safety in an era of global cli mate change, and FAST-S-15-2841 Overflows under specific hydraulic conditions.

Chinnarasri, C. & Tingsanchali, T. & Weesakul, S. & Wong wises, S. 2003. Flow patterns and damage of dike over

topping. International Journal of Sediment Research, Vol.

18, No. 4, 2003, pp. 301-309.

Clopper, P.E. & Chen, Y.H. 1987. Predicting and minimizing

embankment damage due to flood overtopping. In the pro

ceedings of the 1987 National Conference on Hydraulic

Engineering, August 3-7, 1987, Williamsburg, Virginia,

United States, pp. 751-757.

Coleman, S.E. & Andrews, D.P. & Webby, M.G. 2002.

Overtopping breaching of non-cohesive homogeneous

embankments. Journal of Hydraulic Engineering, Vol.

128, No. 9, Sep. 2002, pp. 829-838.

Dupont, E. & Dewals, B.J. & Archambeau, P. & Erpicum, S.

& Pirotton, M. 2007. Experimental and numerical study of

the breaching of an embankment dam. In the proceedings

of the 32nd IAHR Biennial Congress - Harmonizing the

demands from art and nature, 2007, Venice, Italy.

Floods and Reservoir Safety, 1996. Institution of Civil

Engineers, Thomas Telford Publications. Fread, D.L. 1988. Breach:An Erosion Model for Earthen Dam Failures, NWS, Maryland, p.30. Galoie, M. & Zenz, G. 2011. One and two dimensional numerical dam-break flow modeling. In the proceedings of the international symposium on UFRIM (Urban Flood Risk Management), Graz, Austria. Sep. 2011. Pp. 237-243. ISBN: 978-3-85125-173-9. Hanson, G.J. & Temple, D.M. & Cook, K.R. 1999. Dam Overtopping Resistance and Breach Processes Research. In the proceedings of the 1999 Annual Conference of the Association of State Dam Safety Officials, Oct. 10-13, St. Louis, 7 p. Hartung, F. & Scheuerlein, H. 1970. Design of Dam Overflow Rock-fill Dams. In the proceedings of the International Commission on

Large Dams, Tenth International Congress on Large Dams, Q36, R.35, Montreal, Canada, June 1–5, 1970, pp. 587–598. Iooss, B. & Lemaitre, P. 2014. A review on global sensitivity analysis methods. arXiv preprint arXiv:1404.2405. Jandora, J. & Rˇîha, J. 2008. The failure of embankment dams due to overtopping. Brno University of Technology, Faculty of Civil Engineering, Institute of Water Structures, 2008, ISBN: 978-80-214-3527-8. Jun, B. & Oh, K. 1998.Yeonchun Dam Failure and Downstream Dam-break Flood Analysis. Dam Break Modelling, Parallel Session (parallel43), 02.09.1998. Knauss, J. 1979. Computation of maximum discharge at overflow rock fill dams (a comparison of different model test results). In the proceedings of the 13th ICOLD Congress, New Delhi, Q.50–R.9, pp. 143–160. Lemperiere, F. & Courivaud, J.R. & Fry, J.J. 2006. A new analysis of embankment dam failures by overtopping. Commission International Des Grands Barrages, Barcelona, June 2006, pp. 1053–1065. Linford, A. & Saunders, D.H. 1967. A hydraulic investigation of rough and overflow rock fill dams. Report RR 888, British Hydrodynamics Research Association. Løvoll, A. 2006. Breach formation in rock-fill dams' results from Norwegian field tests. Commission International Des Grands Barrages, Barcelona, June 2006, pp. 35–51. Morris, M.W. & Hassan, M. & Vaskinn, K.A. 2007. Breach formation: Field test and laboratory experiments. Journal of Hydraulic Research, Vol. 45, Supplement 1, 2007, pp. 9–17. Roger, S. & Dewals, B.J. & Erpicum, S. & Schwanenberg, D. & Schüttrumpf, H. & Köngeter, J. & Pirotton, M. 2009. Experimental and numerical investigations of dike-break induced flows. Journal of Hydraulic Research, Vol. 47, Issue 3, 2009, pp. 349–359. Schmocker, L. & Hager, W.H. 2009. Modelling dike breaching due to overtopping. Journal of Hydraulic Research, Vol. 47, No. 5, 2009, pp. 585–597. Singh, V.P. 1996. Dam breach modeling technology. Kluwer Academic Publishers. Dordrecht, Netherlands. 1996. pp. 242. ISBN: 0-7923-3925-8. Singh,V.P. & Scarlatros, P. D. 1988.Analysis of Gradual Earth Dam Failure. J. Hydraulic Eng. ASCE, 114(1), 21–42. Wahl, T.L. 1997. Predicting embankment dam breach parameters A needs assessment. In the proceedings of the XXV IAHR Congress, San Francisco, California, August 10–15, pp. 48–53. ISSN: 0074-1477. Wang, Z. & Bowles, D.S. 2006. Three-dimensional noncohesive earthen dam breach model. Part 1: theory and methodology. Advances in Water Resources – ELSEVIER, Vol. 29, Issue 10, October 2006, pp.1528–1545. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Field measurements and numerical modelling on local scour around a ferry slip structure

P. Penchev

Black-Sea Danube Coastal Research Association, Varna, Bulgaria

V. Bojkov

University of Architecture, Civil Engineering and Geodesy, Sofia, Bulgaria

V. Penchev

Black-Sea Danube Coastal Research Association, Varna, Bulgaria

ABSTRACT: This paper reviews a case study on a ferry slip near the town of Nikopol, Bulgaria, located at the Lower Danube River. This ferry slip is an earth and rock-fill hydraulic structure designed to operate as a terminal for passengers and cargo. The location of the ferry slip, together with its complex spatial geometry, led to some structural deformations caused by the complex dynamics of the river flow, the contracted cross section, and the secondary currents along the structure which eventually triggered scour processes at the foot of the structure. Engineering works were done to strengthen the river bed and stop these processes. Detailed hydrographic measurements, including bathymetry, current velocity, turbidity and sediment samples, were carried out to assess the situation of the river bed around the structure in June 2015. Two-dimensional numerical model was elaborated to perform hindcast simulations on the scour mechanism and development.

1 INTRODUCTION

This paper reviews a case study on a recently con structed ferry slip near the town of Nikopol, Bulgaria, located at the Danube River. This ferry slip is an earth and rock-fill hydraulic structure designed to operate as a terminal for passengers and cargo. The ferry slip with a complex spatial geometry was constructed back in 2006 on a very specific site along the Lower Danube. This eventually led to some stability prob lems, caused by the complex dynamics of the river flow, the contracted cross section, the helical flow along the structure, triggering erosion processes at the foot of the structure.

Some structural changes were observed in the ferry slip in 2007, which required detailed hydro graphic measurements including river flow velocities, bathymetry, sediments and geodetic works for the overall assessment of the causes that led to the prob lems with the structure. The measurements were car ried out during low water level (H = 19.39 m), however the observed velocities in the vicinity of the structure were in the range of 0.8–0.9 m/s (at depth z = 0.2 h) and almost 0.3–0.35 m/s near the bottom. The high near bottom velocities and the relatively small sed iment diameter measured as D 50 = 0, 3 mm tend to exceed the critical Shields parameter for initiation of motion of sediment, and therefore massive amounts of sediment were moved away downstream by the river flow

(Daskalov et al. 2007, Penchev et al. 2008). The hydrographic measurements showed a local scour at the foot of the structure of about 3–4 m deep, 100 m long and 30 m wide. After discovering the scour it was decided to put rock-fill mattresses to strengthen the river bed and stop the eroding process. Three rows of Reno mattresses with dimensions 6 × 2 × 0.3 m(L/W/H) were set in place with the assistance of scuba divers. The average rock size is d 50 = 0.1 m. Since then no observations have been made and there have been no reported problems with the structure. Detailed hydrographic measurements were carried out in June 2015 to assess the current situation and to analyze the erosion processes in the river reach. It was discovered that the reinforced with rock-filled mattresses part of the river bed is stable and eventually the scour had seized, however it was found out that the erosion processes are continuing just downstream of this zone and a new scour hole evolved along the structure at its end. This scour does not threaten the ferry slip's stability but in time it will eventually get deeper and must be observed periodically to control the risks for the structure. 2 FIELD MEASUREMENTS Two sets of data from hydrographic measurements were analyzed in this paper – from April 2007 and

Figure 1. Location of the ferry slip structure.

Figure 2. 3D visualization of the ferry slip structure (before

placing the rock-filled mattresses).

June 2015. The results from these measurements are

presented below in the text and further used in the

development of a two-dimensional numerical model

MIKE 21 FM.

The measurements in 2007 revealed the formation

of a potentially dangerous scour in the vicinity of the

ferry slip structure (Daskalov 2007). The scour mag

nitude was estimated to be in the range of 2.5–3.0 m

(see Figure 1).The depth-averaged velocities observed

near the structure were in the range of 0.5–0.6 m/s.

More details about the measured river flow velocities

is given in the table below.

The hydrographic measurements in June 2015 were

carried out by using an unmanned vessel CORES A1

(Penchev et al. 2014), equipped with Echo sounder,

Acoustic DopplerVelocimeter (ADV), D-GPS system,

temperature sensor and water sampling devices. The

area of interest covered 0.25 km 2 . Several sediment

samples from and around the scour hole were taken

and analyzed in geotechnical laboratory. The results

are given in Table 2.

Additional water sampling to assess the turbidity

was carried out. The two probes showed turbidity

μ= 5 ÷ 11 FAU (Formazin Attenuation Units). These

numbers indicate very low turbidity of the river flow

for the time of taking the water samples. Archive data Figure 3. Bathymetry in 2007 (bed levels are in meters in Baltic System). Figure 4. Bathymetry in 2015 (bed levels are in meters in Baltic System); Red line indicates mattresses' location. Table 1. Flow velocities near the ferry slip structure from both measurements. 2007 2015 Velocities (H = 19.4 m) (H = 19.6 m) Surface (0.05 m 0.7–0.85 m/s 0.7–0.8 m/s below surface) Bottom (0.05 m 0.3–0.4 m/s – above bottom) Depth-averaged 0.6–0.7 m/s – Table 2. Characteristic grain size of the sediments. Sediment size [mm] d 16 0,165 d 50 0,310 d 84 0,470 before the construction of Iron Gate I & II are in the range of μ= 100 ÷ 400 g/m 3 (Hydrology Handbook of the Danube River 1977). After the completion of the hydro complex Iron Gate II, the turbidity significantly decreases and is estimated to be μ= 50 ÷ 100 g/m 3 (Penchev 2015). It can be concluded that the river

flow has very low concentration of suspended sedi ments. The morphological changes of the river bed are predominantly caused by bed loads. This was further confirmed by the numerical model and the simulation of the sediment transport.

3 NUMERICAL MODELLING

MIKE by DHI MIKE 21 FM, ST was chosen for the simulation modelling. This numerical model allows the use of a flexible unstructured computational mesh. The modelling system is based on the numerical solution of the two-dimensional shallow water equa tions – the depth-integrated incompressible Reynolds averaged Navier-Stokes equations. Thus, the model consists of continuity, momentum, temperature, salin ity and density equations. In the horizontal domain both Cartesian and spherical coordinates can be used. The spatial discretization of the primitive equations is performed using a cell-centered finite volume method. The spatial domain is discretized by subdivision of the continuum into non-overlapping element/cells. In the horizontal plane an unstructured grid is used comprising of triangles or quadrilateral element. An approximate Riemann solver is used for computation of the convective fluxes, which makes it possible to handle discontinuous solutions. The Sand Transport Module calculates the resulting transport of non cohesive materials based on the flow conditions found in the hydrodynamic calculations.The benefit of using MIKE 21 ST is the possibility of using the helical flow module which is of great importance in river mor phology analysis due to the fact that helical flow is the driving force for bend scour and plays an impor tant role in confluence scour and char build up and migration.

The numerical model is based on the 2007 hydro graphic measurements, aiming to represent the flow conditions and the interaction between the flow and the structure, resulting into the scour and/or accumu lation processes. The two-dimensional model covers an area of 300 × 210 m with a maximum element area of 15 m^2 . The computational mesh is presented below.

The bed resistance is varying in domain, as well as the grain diameter for the sediments. The sediments' properties are of particular interest for this numeri cal model. A special grid was elaborated to represent the different zones with the different sediments and to include the rock-fill mattresses that are modeled as a fixed element. This is done in order to improve the results from the numerical simulations and to properly implement the calculations for the river morphology.

The boundary conditions for the hydrodynamics model are inflow for the west boundary and respectively water level for the north and east boundaries.The inflow is set to 600 m^3/s, corresponding to a water level of 19.50 m. The hydraulic gradient is set to 0.001 to assure smooth transition of the calculations throughout the computational mesh.The parameters and their respective values are listed in Table 3. Figure 5. Computational mesh – triangular elements with an area A ≤ 15 m^2 . Table 3. Simulation parameters for M21 FM model Parameter Value Q_{avg} [m^3/s] 600 H [m] 19.50 Bed resistance [$m^{1/3}/s$] varying in domain (M = 20-40) Grain diameter [mm] varying in domain (d_{50} = 0.3) Time of simulation [days] 30 Time step [s] 1 Figure 6. Velocity field (vector) and water level, point velocities (points indicated). 4 RESULTS FROM NUMERICAL MODELLING The results from the numerical modelling are presented into two groups – hydrodynamics and morphodynamics results. The results from the hydrodynamic simulation are used together with the data from field measurements for the calibration of the numerical model. Below are given the results for the flow velocities and the comparison with the data from field measurements. Figure 6 presents the velocity field in the researched area and give indication of the water level before and after the structure.

Table 4. Measured and modeled flow velocities. Surface Averaged Model

Point No	velocity 2015	velocity 2007	velocity
1	0.90 m/s	0.64 m/s	0.67 m/s
2	0.84 m/s	0.63 m/s	0.60 m/s
3	0.60 m/s	0.48 m/s	0.54 m/s

The depth-averaged velocities are in the range 0.5–0.7 m/s which corresponds to the velocities measured back in 2007 and 2015 (hydraulic conditions were almost identical). In Table 4 is given the comparison between measured and modeled data.

The calculated water level in the reach is also of interest since it gives a clear view on the backwater effect just before the structure (up to 2-3 cm higher water level).

The comparison of the measured and modeled flow velocities shows good match and also indicates that the morphological changes in the reach are balanced.

The morphological changes observed in the numer ical model showed similar pattern when compared to the data from the hydrographic measurements in 2015. The same simulation time covering a time period of 30 days was used. The boundary conditions were same as for the hydrodynamics calculation; however two additional features are used – suspended concentra tion and helical flow. The helical flow is the main force for bend erosion. It is considered to have a significant effect in the researched river reach because of the com plex interaction between the river flow and the spatial structure. In general the results can be regarded as qual itative and not quantitative. This means that in this paper the zones where erosion or accumulation pro cesses occur are only indicated. The mechanism and the complex relation between the flow velocities and the development of helical flow around the structure have been analyzed. The results are presented below on the figure and are given in terms of the rate of bed level change. This is done because of the numerical modelling approach to use a steady state condition – average discharge and respective water levels in the researched reach. The rate of bed level change is in the magnitude of ±0.1–0.2 cm/day for the simulated hydraulic conditions.

The results from the numerical modelling are con cerning because of the clear trend of constant erosion in the zone even by simulating an average discharge. Furthermore the turbidity of the Danube River in the Bulgarian reach is quite low and sedimentation of sus pended particles is not significant. The changes in the river bed are significant and alter the reach in a way making it deeper which changes the flow velocity distribution but also indicates sedimentation down stream of Nikopol which could result in sand bars and difficulties for the navigation.

The different zones of erosion/accumulation are further analyzed by using the data from the hydro

graphic measurements in 2015 and are post processed Figure 7. Example rate of bed level change (black – accumulation, grey – erosion). in GIS environment. The outputs are indication of the zones where the river bed had eroded or aggregated and quantitative evaluation. The patterns of the erosion/accumulation derived by the numerical model are also compared to the data from the hydrographic measurements. 5 GIS BASED SCOUR EVALUATION A time saving approach to evaluate the scour magnitude around the ferry slip has been reviewed in this paper. The method uses the open source QuantumGIS (QGIS) platform and is focused on the elaboration of 3D high resolution raster images from measurements data and intersecting them to get the total volume of eroded material and to indicate the places where there is a scour or accumulation. This approach can be used on a regular basis with data from numerical modelling to assess the trends in erosion/accumulation at different periods of time. At the same time it is useful for the proper visualization and calculations for the engineering works that has to be executed depending on the identified problems. It can be used also to calculate the total volume of the morphological changes (both erosion and accumulation). Two identical triangular irregular networks (TIN) have been created for both hydrographic measurements in 2007 and 2015.The cell size for both networks is 1 × 1 m. The approach includes the subtraction of the two raster images in 2 different resulting raster images – one for the eroded zones (positive values) and one for the sediment zones (negative values). The algorithm is based on the Raster Calculator function in QGIS and the steps are as follows: (a) Erosion The resulting TIN is created by subtracting the referent TIN (Tiff2007) with the one from the last measurements Tiff2015, by selecting all positive values that represent the erosion.

Figure 8. Morphological changes evaluation area

(polygon).

(b) Accumulation

The resulting TIN is created by subtracting the referent

TIN (Tiff2007) with the one from the last measure

ments Tiff2015, by selecting all negative values that

represent the accumulation.

The above two steps give as an output two differ

ent TINs, one for erosion and one for accumulation.

The volume for the erosion and accumulation respec

tively can be calculated by using the function Zonal

Statistics. It allows the newly created raster image (Er_Tiff/Acc_Tiff) to be analyzed in terms of total area, area with positive values or negative values. The output of the function is a shapefile containing three columns with a predetermined prefix.The first column is the sum of all cells, the second column is the sum of all positive or negative values, and the third column is the ratio between the first and the second column. It must be pointed out that these calculations are made with a cell size of 1 × 1 m. In cases when different cell size (dx × dy) is used, it must be recalculated with the result of the cell area (dx × dy).

In the particular case of the researched river reach the scope of the data is not equal, so a polygon is cre ated to cover the area of interest. The data from the hydrographic measurements in 2007 and 2015 covers different areas and it was decided to use a polygon that covers the area where both bathymetric measurements overlap. The polygon covers an area of 0.035 km^2 (35 000 m^2) and is shown below on the figure.

The resulting raster image is shown below with the indicated zones for erosion and accumulation. The overall erosion in the researched zone was estimated to be around 26 500 m^3 , while the accumulation around 10 500 m^3 . Due to the relative small scope of data measurements from 2007 this technique is used.

The maximum scour depth is 3.25 m and the maxi

mum observed accumulation is 2.70 m.These numbers

indicate significant changes in the river bed and it

is recommended that the situation must be moni

tored constantly, so that the stability of the structure

is assured. The results from the numerical modelling Figure 9. GIS visualization of the scour development in 2015 (black polygons). Figure 10. GIS visualization of the accumulation development in 2015 (black polygons). also show this trend for the development of erosion downstream of the structure. The magnitude of the erosion and accumulation calculated by using this GIS approach showed very good match with the results from the numerical modelling, where the average rate of erosion during steady state simulation was estimated to be in the range of 1-2 cm/day. Simple calculation for the whole period between 2007 and 2015 gives a rough value of approximately 3.0 m. This GIS-based approach to indicate the changes of the river bed and in the same time to calculate the volumes of these changes is very useful and can be applied on a regular basis for a researched river reach. The process can be easily automated by using python scripts and assigning them as executable tasks. The processed data clearly shows how the rockfilled mattresses stopped all erosion processes at the foot of the ferry slip. On the other hand it must be stated that erosion processes are ongoing towards the thalweg, covering a broader reach. 6 CONCLUSIONS Detailed hydrographic survey and numerical modelling have been used to hindcast the morphological changes in the river bed around the ferry slip at Nikopol, located in the Lower Danube.

Hydrographic measurements were carried out in

June 2015. Data for the bathymetry, flow velocities and

sediments were obtained. The current situation was

assessed and it was found out that the reinforced parts

of the river bed have strengthened the river bed and

no significant morphological changes were observed in this zone. Data is compared to previous measurements that were executed in 2007, which are used for the establishment of a two-dimensional hydrodynamic numerical model.

DHI's MIKE 21 Flow Model, for both hydrodynamic and morphological computations, was used to simulate hydraulic and morphological processes in the investigated area. The results from hydrographic surveys were used to calibrate and verify the model. Numerical model simulations carried out within the present study showed good correspondence to the data collected during the field measurements in 2015. The erosion rate is computed to be approximately 1 cm/day during steady state simulation with an average discharge (Q_{avg}). The accumulation rate is also estimated to be in the same magnitude.

Such numerical model can be used for long-term simulations for future forecasts of the dynamics of the river bed in this zone.

The calculation of the overall erosion/accumulation has been made by using GIS tools. This approach gives the possibility of visualization and in the same time actual calculation of the volumes of erosion/accumulation.

The results obtained provide a good ground for further use of the developed numerical model, in com

bination with the demonstrated GIS approach, for

future forecasts of the dynamic of the river bed in

Experimental and numerical study of scour downstream Toachi Dam

Luis G. Castillo

Hidr@m Group, Civil Engineering Department, Universidad Politécnica de Cartagena, Spain

Marco Castro

Centro de Investigaciones y Estudios en Recursos Hídricos (CIERHI), Escuela Politécnica Nacional, Ecuador

José M. Carrillo

Hidr@m Group, Civil Engineering Department, Universidad Politécnica de Cartagena, Spain

Daniel Hermosa, Ximena Hidalgo & Patricio Ortega

Centro de Investigaciones y Estudios en Recursos Hídricos (CIERHI), Escuela Politécnica Nacional, Ecuador

ABSTRACT: The study analyzes the expected changes in the Toachi River (Ecuador) as a result of the con

struction of the Toachi Dam (owned by CELEC EP Hidrotoapi). Toachi is a concrete dam with a maximum

height of 59 m to the foundations. The top level is located at an altitude of 973.00 meters above sea level. With

normal maximum water level located at 970.00 m, the reservoir has a length of 1.30 km in the Sarapullo River

and 3.20 km in the Toachi River. The dam has a free surface weir controlled by two radial gates. It consists in

2 channels located in center of the dam that end in ski jump. The discharge is controlled by radial gates in order

to ensure the accurate operation in the event when the gates are partially open. The spillway has been designed

to spill up to a rate flow of 1213 m^3/s. It is necessary to know the shape and dimensions of the scour generated downstream of the dam. This scour is studied with four complementary procedures: laboratory model with 1:50 Froude scale similitude, empirical formulae obtained in models and prototypes, semi-empirical methodology based on pressure fluctuations-erodibility index, and Computational Fluid Dynamics (CFD) simulations.

1 DAM CHARACTERISTICS

The Toachi Dam is located in the South-West of the Quito city (Ecuador). It is a concrete dam with a max imum height of 59 m to the foundations. The top level has a length of 170.5 m and 10 m of thickness. It is located at an altitude of 973 meters above the sea level.The upstream and downstream embankment side slopes are 0.3/1.0 and 0.7/1.0 (horizontal/vertical), respectively.

The reservoir collects water from the basins of the Toachi and Sarapullo rivers. It has a total volume of 8 Hm^3 with normal maximum water level located at 973 meters. At this level, the reservoirs have a length 1.3 km in the Sarapullo River and 3.2 km in the Toachi River.

The dam has two Creager spillways controlled by gates. The spillways end in a ski jump and they have two baffles to divide the flow.The design flow matches a 1000 years return period (1213 m^3/s) with an energy head of 7.50 m. There are two bottom outlets whose capacity is 800 m^3/s. The dam also has a stepped spill way for the Sarapullo River with a design flow of 40 m^3/s (Figure 1). Figure 1. Three-dimensional view and physical model of the Toachi Dam. 2 PHYSICAL MODEL The physical model was built with a Froude scale 1:50 in the Centro de Investigaciones y Estudios en Recursos Hídricos (CIERHI) of the Escuela Politécnica Nacional (Ecuador). The scour downstream the dam was analyzed by using different flows according

Figure 2. Physical model of the Toachi ski jump.

Table 1. Rate flows and maximum scour depths in the physical model with d_{model} = 0.020 m ($d_{prototype}$ = 1.00 m).

Horizontal distances D from the dam to the maximum scour depth.

Q_{model}	$Q_{protype}$	$Y_{s\ model}$	$Y_{s\ protopype}$	D_{model}	$D_{prototype}$
(l/s)	(m^3/s)	(m)	(m)	(m)	(m)
14.38	254	0.131	6.57	1.035	51.75
28.26	500	0.161	8.05	1.219	60.95
40.21	711	0.141	7.05	1.282	64.10
56.51	999	0.143	7.15	1.233	61.65
68.63	1213	0.133	6.65	1.284	64.20

to the Hydrology Inform of the Toachi-Pilaton Dam (Hidrotoapi, 2010).

The river bed (Figure 2) was modeled considering three uniform gravels sizes, whose mean value were d_{model} = 0.020, 0.015 and 0.010 m in scale model.

Table 1 summarizes the maximum scour depth below the original bed (Y_s) and the distance from the dam to the maximum scour (D) for the 1.00 m gravel size (0.020 m in model). The maximum scour Y_s = 7.15 m was obtained for the design flow of 999 m^3/s, reducing the scour to 6.65 m in the bigger tested flow. The maximum distance of the scour 64.20 m was obtained with the maximum flow.

3 EMPIRICAL FORMULAE

In the study, 30 formulae are examined. The scour hole is estimated for flows of several return periods. Most of the equations were obtained by dimensional and statistics analysis of data obtained in Froude scale reduced models, with few formulae based on proto types and many obtained for the ski-jump. The general expression is:

where Y_0 is the tailwater depth, k an experimental coef ficient, q the specific flow, H the energy head, g the gravity acceleration, d_m the average particle size of the bed material, d_{85} the bed material size in which 85% is smaller in weight, and d_{90} the bed material size in which 90% is smaller in weight. The rest of variables are showed in Figure 3. Figure 3. Scheme of scour in Toachi Dam. Figure 4 shows the results obtained with the 30 formulae considering the sediment size of 1.00 m. The mean value +/-1 standard deviation is indicated. After removing the formulae whose values fall out of the +/-1 standard deviation threshold. Figure 5 shows the mean value +/- 0.50 standard deviation values obtained, together with the scale model results. If the mean value for the design flow (1213 m^3/s) were considered, the scour would reach a depth of 7.81 m. However, if the mean value +0.50 standard deviation was taken into account, then the same flow would scour 13.68 m. Table 2 shows four of the general expressions whose values are closer to the mean value, while Table 3 shows the coefficients corresponding to four simplified formulae with values in the same range. In Figure 5, the values obtained in the scale model are close similar to the mean values calculated. We can observe that all values obtained in the physical model fall in the mean value +/- 0.50 standard deviation. 4 SEMI-EMPIRICAL FORMULAE The erodibility index is based on an erosive threshold that relates the magnitude of relative erosion capacity of water and the relative capacity of a material (natural or artificial) to resisting scour. There is a correlation between the stream power or magnitude of the erosive capacity of water (P) and a mathematical function [f (K)] that represents the relative capacity of the material to resist erosion. On the erosion threshold, this may be expressed by the relationship P = f (K). If P > f (K), with the erosion threshold being exceeded and the material eroded. Scour in turbulent flow is not a shear process. It is caused by turbulent and fluctuating pressures (Annandale, 2006). Quantification of pressure fluctuations of incident jets in stilling basins have been studied mainly by Ervine and Falvey (1987), Ervine et al. (1997), Castillo (1989, 2002, 2006, 2007), Castillo et al. (1991, 2007), Puertas (1994), Bollaert (2002), Bollaert and Schleiss (2003), Melo et al. (2006), Felderspiel (2011), Carrillo (2014), and Castillo et al. (2014, 2015). The dynamic pressures of jets are a function of the turbulence intensity at the discharge conditions, length

Figure 4. Scour of the ski jump obtained with 30 formulae and the threshold of +/- 1 standard deviation.

Figure 5. Scour of the ski jump obtained with the formulae in the threshold of +/- 0.50 standard deviation. Table 2. Four scour general formulae with values that fall in the mean value +/- 1 standard deviation. Author Year Equation Jaeger 1939 D S = 0.6q 0.5 H 0.25 (h/d m) 0.333 Martins-A 1973 { D S = 0.14N - 0.73 h 2 N + 1.7h. N = (Q 3 H 1.5 /d 2 m) 1/7 Veronese modified 1994 D S = 1.90h 0.225 q 0.54 sin θ T Bombardelli & Gioia 2006 D S = K q 0.67 H 0.67 g 0.33 d 0.33 (ρ ρ s - ρ)

Table 3. Coefficients of five scour simplified formulae

with values that fall in the mean value +/- 1 standard deviation.

Author	Year	k	a	b	c	d	e	f	h	i
Tairamovich	1978	0.633	0.67	0.25	0	0	0	0	0	0
Martins-B	1975	1.50	0.60	0.10	0	0	0	0	0	0
Mason-Arumugam A	1985	3.27	0.60	0.05	0.15	0	0.30	0.10	0	0
Damle-C	1966	0.362	0.50	0.50	0.50	0	0	0	0	0
INCYTH	1981	1.413	0.50	0.25	0	0	0	0	0	0

Table 4. Erodibility index parameters (Adapted from Annandale, 2006).

of the jet flight, diameter (circular jet) or thickness (rectangular jet) in impingement jet conditions and water cushion depth.

Annandale (1995, 2006) summarized and established a relationship between the stream power and the erodibility index for a wide variety of materials and flow conditions. The stream power per unit of area available of an impingement jet is:

where γ is the specific weight of water, Q the flow, H the drop height, and A the jet area on the impact surface. The jet area was estimated using the equations of the impingement jet thickness for the free falling jet case (Castillo et al., 2014b and 2015b), in which the throwing distance and the specific flow are considered. The impingement jet thickness formula has been obtained as:

where B_g is the thickness due to gravity effect, ξ

the jet lateral spread distance due to the turbulence effect, q the specific flow, H the fall height, and h is the energy head at the crest weir. $\phi = K_\phi T_u$, being T_u the turbulence intensity, and K_ϕ an experimental parameter (1.14 for circular jets and 1.24 for the three-dimensional nappe flow case). The erodibility index (Annandale, 2006) is defined as: being Ms the number of resistance of the mass, K_b the number of the block size, K_d the number of resistance to shear strength on the discontinuity contour, and J_s the number of structure relative of the grain. Table 4 shows the formulae of the parameters. The threshold of rock strength to the stream power, expressed in kW/m^2, is calculated and based on the erodibility index K. The dynamic pressure on the bottom of the stilling basin is based on two components: the mean dynamic

Figure 6. Reduction factor F of fluctuating dynamic pres

sure coefficient.

Figure 7. Mean dynamic pressure coefficient, C_p, for the

nappe (rectangular) case.

pressure (C_p) and the fluctuating dynamic pressure

(C'_p). These dynamic pressure coefficients are used as

estimators of the stream power reduction coefficients,

by an effect of the jet disintegration in the air and

their diffusion in the stilling basin (Annandale, 2006).

Hence, the dynamic pressures are also a function of the

fall height to disintegration height ratio (H/L_b) and

water cushion to impingement jet thickness (Y/B_j).

The total dynamic pressure is:

where $C_p(Y/B_j)$ is the mean dynamic pressure coef

ficient, $C'_p(Y/B_j)$ the fluctuating dynamic pressure

coefficient, P_{jet} the stream power per unit of area, and F the reduction factor of the fluctuating dynamic pressure coefficient.

In the rectangular jet case, Carrillo (2014) and Castillo et al. (2014, 2015) adjusted the formulae by using new laboratory data (Figures 6, 7 and 8).

Table 5 shows the results obtained in the three types of materials. Table 6 lists the results of incident stream (P_{jet}) and diffusion ($P_{jet}/Y/B_j$) jet power.

Figures 9 and 10 show the stream power of the jet, together with the power threshold for the three different materials. We can observe that all flow rates impin

gent with enough power stream to erode a material Figure 8. Fluctuanting dynamic pressure coefficient, C'_p, for the nappe flow case. Table 5. Parameters of three types of materials. Material Type Parameters I II III d_{50} (m) = 0.50 0.74 1.02 d_{84} (m) = 0.63 0.88 1.20 θ ∘ = 32.00 33.00 34.00 M_s = 0.35 0.37 0.40 K_b = 244.14 681.47 1728.00 K_d = 0.62 0.65 0.67 J_s = 1.00 1.00 0.70 K = 53.39 163.75 326.36 P_{rock} (kW/m ^{2}) = 19.75 45.77 76.78 Table 6. Final water cushion ($Y_0 + Y_s$), scour (Y_s), initial water cushion (Y_0), incident stream power (P_{jet}) and reduced stream power by diffusion [P_{jet} (Y/B_j)]. $Y_0 + Y_s$ Y_s Y_0 P_{jet} P_{jet} (Y/B_j) Q (m) (m) (m) (kW/m ^{2}) (kW/m ^{2}) 254 12.05 6.57 5.48 76.94 3.63 500 14.15 8.05 6.10 94.26 19.79 711 15.73 7.05 8.68 101.59 43.60 1000 17.10 7.15 9.95 113.02 71.50 1213 18.65 6.65 12.00 108.31 64.59 Figure 9. Incident stream power P_{jet} and reduced stream power by diffusion P_{jet} (Y/B_j) of the jet. Power threshold of three types of materials (I, II, and III).

Figure 10. Stream power of the jet for different flows as a function of the erodibility. Three types of materials (I, II and III). Values ($Y_0 + Y_s$) are variables in each flow (see Table 6).

with power threshold of 76.78 kW/m ^{2}. However, the reduced stream power by diffusion (254 and 500 m^3/s), due to the effect of the water cushion ($Y_0 + Y_s$) stablished in the model, are below the power threshold of 19.75 kW/m^2. The flow 711 m^3/s does not have enough power to erode the material power threshold of 45.77 kW/m^2. Flow rates of 999 and 1213 m^3/s no longer have the capacity to erode the material power threshold of 76.78 kW/m^2.

5 NUMERICAL SIMULATIONS

As a complement of the empirical and semi empirical methodologies, three-dimensional mathematical model simulations were carried out. These programs allow a more detailed characterization than one-dimensional and two-dimensional numerical models and, thus, a detailed study of local effects of the sediments transport. The numerical simulation of the hydraulic behavior and scour by the action of the ski jump was analyzed.

The computational fluid dynamics (CFD) program FLOW-3D v11.1 was used. This program solves the Navier-Stokes equations using finite differences. It incorporates various turbulence models, a sediment transport model and an empirical model bed erosion (Guo, 2002; Mastbergen and Von den Berg, 2003; Brethour and Burnham, 2011), together with a method for calculating the free surface of the fluid without solving the air component (Hirt and Nichols, 1981). Pressures obtained in the stagnation point and their associated mean dynamic pressure coefficients were compared with the parametric methodology proposed by Castillo et al. (2013, 2014). This methodology was successfully used to estimate the scour downstream Paute-Cardenillo Dam (Castillo and Carrillo, 2016). In order to simulate the proper functioning of the ski jump, several simulations were carried out by means of sensibility analysis: air entrainment mod els, turbulence models, grid size and type of solver, among others. Simulations were performed at labora tory scale. Multiple mesh blocks were used to solve the problem. The spillway and the ski jump were solved Figure 11. Numerical simulation of the scour downstream Toachi Dam. Figure 12. Longitudinal and transversal scour shape measured and simulated. with a mesh size of 0.005 m, while the reservoir and the plunge pool were resolved with a mesh size of 0.02 m. In the sediment scour model, the critical Shields number was calculated using Soulsby-Whitehouse equation, while the Meyer-Peter & Müller equation was used to compute the bed load transport rate. Two bed load coefficients for low sediment transport (β = 5.0 and β = 6.5) and the maximum packing fraction were used to calibrate the model. Figure 11 shows the results obtained for the design flow (Q = 1213 m^3/s) and considering a grain size of 1.00 m. The maximum scour depths were 8.50 m and 7.50 m, for β values of 6.5 and 5.0, respectively. These values are a bit bigger than the value obtained in the physical model 6.65 m and around the mean value obtained with the empirical formulae 7.81 m. Figure 12 compares the scour shape observed in laboratory with the numerical simulation in the planes in which the maximum scour value was measured.

The horizontal distances from the dam to the maxi mum scour depth were 61.50 m (β = 6.5) and 63.50 m (β = 5.0), similar to the value obtained in laboratory of 64.20 m.The longitudinal scour length from the lab oratory data was around 51.55 m while the simulated value was 49 m. The transversal scour length was near the complete transversal section in both cases.

The main differences in the scour measured in lab oratory and calculated seem to be related to the fact that the current version of FLOW-3D does not allow to activate the density evaluation and drift-flux models in the air entrainment model when the sediment scour model is used. This generates impact jets more com pact that if the air entrainment mechanism were solved in the correct way.

6 CONCLUSIONS

In this paper, similar results have been obtained by solving the problem from four different perspec tives: physical model, empirical formulations, ero sion potential semi-empirical formulation and CFD simulations.

The results obtained with the reduced model and the numerical simulation with CFD, have been obtained in scale laboratory. To extrapolate to prototype, the designer should take the scale effects into account.

As the air entrainment is not correctly reproduced at

laboratory scales, these results are on the safe side.

The results demonstrate the suitability of cross

ing methodologies to solve complex phenomena.

Thus, numerical simulations were used to comple

ment the classical formulations and the laboratory

results, allowing a better understanding of the physical

phenomena in order to obtain an adequate solution.

ACKNOWLEDGMENTS

We are grateful to CELEC EP HIDROTOAPI for

the data provided. The third author acknowledges the

support of Escuela Politécnica Nacional (Ecuador)

through the program Invited professor, and the support

of the program PMPDI-UPCT 2015, which allowed

him to develop a stay at the Centro de Investigaciones y

Estudios en Recursos Hídricos CIERHI of the Escuela

Politécnica Nacional (Ecuador), from September to

December, 2015.

Annandale, G. W. 1995. Erodibility. Journal of Hydraulic

Research 33 (4), 471–494.

Annandale, G. W. 2006. Scour Technology. Mechanics and

Engineering Practice. McGraw-Hill.

Bisaz, E. &Tschopp, J. 1972. Profundidad de erosion al pie
de

un vertedero para la aplicación de correccion de arroyos en quebradas empinadas. Proceedings of the Fifth Congreso

Latinoamericano de Hidraulica (IAHR), Lima, Peru, 447-

456 (in Portuguese). Bollaert, E. 2002.TransientWater Pressures in Joint and Formations of Rock Scour due to High-Velocity Jet Impact, Communication N o . 13. Laboratory of Hydraulic Constructions, École Polytechnique Fédérale de Lausanne, Switzerland. Bollaert, E.F., & Schleiss, A. 2003. Scour of rock due to the impact of plunging high velocity jets. Part 1: A stateof-the-art review. Journal of Hydraulic Research 41(5), 451–464. Bombardelli, F.A. & Gioia, G. 2006. Scouring of granular beds by jet-driven axisymmetric turbulent cauldrons. Phys. Fluids 18(8), 088–101. Brethour, J., & Burnham, J. 2010. Modeling Sediment Erosion and Deposition with the FLOW-3D Sedimentation & Scour Model. Flow ScienceTechnical Note, FSI-10-TN85, 1–22. Carrillo, J. M. 2014. Metodología numérica y experimental para el diseño de los cuencos de disipación en el sobrevertido de presas de fábrica. PhD Thesis. Universidad Politécnica de Cartagena, Spain (in Spanish). Castillo, L. 1989. Metodología experimental y numérica para la caracterización del campo de presiones en los disipadores de energía hidráulica. Aplicación al vertido en presas bóveda. PhD Thesis. Universitat Politécnica de Catalunya. Barcelona, Spain (in Spanish). Castillo, L., Dolz, J. & Polo, J. 1991. Acquisition and analysis of data to characterize dynamic Actions in Hydraulic Energy Dissipators. XXIV IAHR Congress. D, 273–280. Madrid. Castillo, L. 2002. ParametricalAnalysis of the Ultimate Scour and Mean Dynamic Pressures at Plunge Pools. Workshop on Rock Scour due to High Velocity Jets. École Polytechnique Fédérale de Lausanne. Castillo, L. 2006. Areated jets and pressure fluctuation in plunge pools.The 7th International Conference on Hydroscience and Engineering (ICHE-2006), IAHR, ASCE, Drexel University. College of Engineering. DSpace Digital Library. DU Haggerty Library, Philadelphia, USA. Castillo, L., Puertas, J. & Dolz, J. 2007. Discussion about Scour of rock due to the impact of plunging high velocity jets Part I: A state-of-the-art review. Journal of Hydraulic Research 45 (6), 853–858. Castillo, L. 2007. Pressure characterization of undeveloped and developed jets in shallow and deep pool. 32nd Congress of IAHR, the International Association of Hydraulic Engineering & Research, 2, 645–655, Venice, Italy. Castillo, L.G., & Carrillo, J.M. 2012. Hydrodynamics characterization in plunge pool. Simulation with CFD methodology and validation with experimental measurements. 2nd IAHR Europe Congress, Munich. Castillo, L.G., & Carrillo, J.M. 2013. Analysis of the ratio scale in nappe

flow case by means of CFD numerical simulations. Proceedings of 2013 IAHR Congress, Chengdu, China. Castillo, L.G., & Carrillo, J.M. 2014a. Scour analysis downstream of Paute-Cardenillo Dam. 3rd IAHR Europe Congress, Porto. Castillo, L.G., Carrillo, J.M. & Sordo-Ward, A. 2014b. Simulation of overflow nappe impingement jets. Journal of Hydroinformatics 16(4), 922–940.DOI: 10.2166/hydro. 2014.109. Castillo, L.G., & Carrillo, J.M. 2015a. Characterization of the dynamic actions and scour estimation downstream of a dam. Dam Protections against Overtopping and Accidental Leakage. CRC Press, 231–243. DOI: 10.1201/ b18292-26.

Castillo, L.G., Carrillo, J.M. & Blázquez, A. 2015b. Plunge pool dynamic pressures: A temporal analysis in the nappe flow case. Journal of Hydraulic Research 53(1), 101–118. DOI: 10.1080/00221686.2014.968226.

Castillo, L.G., Carrillo, J.M. & Álvarez, M.A. 2015c. Com plementary Methods for Determining the Sedimentation and Flushing in a Reservoir. Journal of Hydraulic Engi neering, 141(11):05015004.DOI: 10.1061/(ASCE)HY. 1943-7900.0001050.

Castillo, L.G. & Carrillo. 2016. Scour, Velocities and Pres sures Evaluations Produced by Spillway and Outlets of Dam. Water, 8 (3), 68. DOI: 10.3390/w8030068.

Chee, S.P. & Padiyar, P.V. 1969. Erosion at the base of flip buckets. Engineering Journal, Inst. of Canada 52(111), 22–24.

Ervine, D. A., & Falvey, H.R. 1987. Behavior of turbulent jets in the atmosphere and plunge pools. Proceedings of the Institutions of Civil Engineers, 83 (2), 295–314.

Ervine, D. A., Falvey, H.R. & Whiters, W. 1997. Pressure

Fluctuations on Plunge Pool Floors. Journal of Hydraulic Research 35 (2).

Escuela Politécnica Nacional, HIDROTOAPI E.P. 2013. Estudio experimental en modelo hidráulico. Escala 1:50. Verificación experimental del diseño definitivo de la presa Toachi y obras complementarias. InformeTécnico Fase IV. Quito, Ecuador.

Felderspiel, M.P. 2011. Response of an embedded block impacted by high-velocity jets. PhD Thesis. École Poly technique Fédérale de Lausanne, Suisse.

Flow Sciences Inc. 2015. FLOW-3D Users Manual Version 11.1. Santa Fe, New Mexico.

Guo, J. 2002. Hunter Rouse and Shields diagram. Proc 1th IAHR-APD Congress, Singapore, 2, 1069–1098.

Hartung, W. 1959. Die Kolkbildung hinter Uberstromen wehren im Hinblick auf eine beweglich Sturzbettgestal tung. Die Wasser Wirtschaft 49(1), 309–313 (in German).

Hidrotoapi E.P., 2010. Informe 6256.0-R-18. Presa Toachi. Hidráulica. Memoria de cálculo. Quito.

Hirt, C. W. & Nichols, B. D. 1981. Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries. Journal of Computational Physics 39 (201). Jaeger, C. 1939. Uber die Aehnlichkeit bei flussaulichen Modellversuchen. Wasserkraft und Wasserwirtschaft, 34(23/24), 269 (In German). Machado, L.I. 1980. Formulas to calculate the scour limit on granular or rock beds. XIII National workshop on large dams, Subject 1, Rio de Janeiro, Brazil, 35–52. Martins, R. 1975. Scouring of rocky riverbeds by free-jet spillways.

Water Power & Dam Construction. April, 1975. Mason, P.J. 1989. Effects of air entrainment on plunge pool scour. Journal of Hydraulic Engineering 115(3), 385–399. Mason, P.J. & Arumugan, K. 1985. Free Jets Scour below Dams and Flip Buckets, Journal of Hydraulic Engineering 111(2), 220–235. Mastbergen, D. R. & Von den Berg J. H. 2003. Breaching in fine sands and the generation of sustained turbidity currents in submarine canyons. Sedimentology 50, 625–637. Melo, J. F., Pinheiro, A. N. & Ramos, C. M. 2006. Forces on plunge pool slabs: influence of joints location and width. Journal of Hydraulic Engineering 132 (1), 49–60. Meyer-Peter, E. & Müller, R. 1948. Formulas for bed-load transport. Proceedings of the 2nd Meeting of the International Association for Hydraulic Structures Research, 39–64. Mirtskhulava, T. E. 1967. Alguns Problemas da Erosao nos Leitos dos Rios. Moscow. Trans. No 443do L.N.E.C. (in Portuguese). Puertas, J. 1994. Criterios hidráulicos para el diseño de cuencos de disipación de energía en presas bóveda con vertido libre por coronación. PhDThesis. Universidad Politécnica de Cataluña, Spain (in Spanish). Rubinstein, G.L. 1963. Laboratory investigation of local erosion on channel beds below high overflow dams. Transactions of Coordination Conferences on Hydraulic Engineering. 1ss. VII, Conference on Hydraulics of High Head Water Discharge Structures. Gosenergoizdat M.L. Taraimovich, I. I. 1978. Deformation of channels below high head spillways on rock foundations. Hydrotechnical Construction 9, 917–922. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Design of scour protections and structural reliability techniques

Tiago Fazeres-Ferradosa, Francisco Taveira-Pinto & Luciana das Neves

Faculty of Engineering, University of Porto, Portugal

Maria Teresa Reis

National Laboratory for Civil Engineering, Portugal

ABSTRACT: In order to account for scour problems, protection systems are usually employed around the

foundations, i.e. Arklow Bank OWF and Scroby Sands OWF (Whitehouse et al., 2011). Their design is mainly

of semi empiric nature (De Vos et al., 2011) and probabilistic approaches haven't yet reached a mature state of development, for cases of waves and currents combined. This paper provides the preliminary results of a reliability based methodology for the design of rip-rap systems, around a monopile foundation. The physical modelling of scour phenomena was combined with Monte Carlo Method in order to evaluate the performance of the protection. A failure criteria and the respective probabilities of failure are associated with the safety factor concept, by means of the shear stress quantification over the top layer blocks.

1 INTRODUCTION

The renewable energy market is growing, according to the most recent statistics of EWEA, around 2342 MW of offshore wind capacity were installed, during the first half of 2015 (EWEA, 2015).

Monopiles are commonly employed as part of the foundations for offshore wind turbines (Sørensen & Ibsen, 2013). The design of such substructures is often conservative and there is still margin for an improvement of cost benefit ratios.

The support structure can represent about 25-34% of the total cost of an offshore wind turbine. Therefore, the economic feasibility of these infra-structures must be achieved by means of low cost and low risk concepts (LeBlanc, 2009; Bhattacharya, 2014).

One of the major problems affecting the design of monopiles is the scour phenomena, which can affect the stability of the structure, namely reducing their ulti mate capacity and the dynamic response (Prendergast et al., 2015), eventually leading to collapse.

In order to account for scour problems, protection systems are usually employed around the founda tions, i.e. Arklow Bank OWF and Scroby Sands OWF (Whitehouse et al., 2011).

Their design is mainly of semi empiric nature (De Vos et al., 2011) and probabilistic approaches haven't yet reached a mature state of development, for cases of waves and currents combined.

Reliability-based methodologies imply the calcula tion of failure probabilities, which enable to provide a measure of risk instead of the typical safety fac tor concept. Reliability techniques have been mainly developed as structural design methodologies, typically applied in structural elements, such as piles, beams and other elements of buildings design, e.g. Das & Zhang (2003), Baecher & Christian (2003) or Cremona (2011). Some works have extended the reliability methodology to scour phenomena in fluvial conditions, as Johnson (1992), Bolduc (2006), Muzzammil & Siddiqui (2009) or Briaud et al. (2014). However, the lack methodologies for the reliability-based design of scour protections in offshore environment is evident. The present work aims to provide the preliminary results of a reliability based methodology, to design rip-rap systems, around a monopile foundations in marine environment.This method was based on Monte Carlo Method coupled with the Latin HypercubeAlgorithm, to minimise the number of simulations needed for the probabilities' convergence. 2 METHODOLOGY 2.1 Database The present study concerns the database presented in De Vos (2011), which compiled a series of scour protection tests around a monopile foundation. The protections were tested under typical marine conditions were currents and waves are combined. The tests were performed in a flume with the following dimensions: 30 m in length, 1 m in width and 1.2 m in height.

Figure 1. Scour protection system and dimensions, without

the pile placed (De Vos, 2011).

Table 1. Probability distribution functions and variables

considered for Monte Carlo Simulations.

Input Parameter pdfs

Water depth (d) Normal Dist.

Current velocity (U_c) Normal Dist.

Significant wave height (H_s) Normal Dist.

The protection consisted in a rip-rap layer covering

a filter one, as shown in the following figure (D = pile

diameter).

The details of physical modelling results and labo

ratorial conditions are founded in De Vos (2011).

The mentioned database resulted in a best fit model

equation, which can be used to assess shear stress on

the top layer of the protection, at a 1/1 scale (prototype

scale). The equation is defined as follows:

Equation (1) depends on the shear stress generated

by currents, τ_c, and the one generated by the waves'

action, τ_c. Both variables are obtained within the same

procedure used in the upper mentioned author. Equa

tion (1) provides an estimate of the critical shear stress

for a certain sea state condition.

2.2 Basic Random Variables

The basic random variables considered, and the pre

defined probability distribution functions (pdfs) are

presented in table 1:

Normal distribution was assumed since according

to previous authors, Chang (1994) and Muzzammil &

Siddiqui (2009), this yields a conservative estimate of

the probabilities of failure.

No specific study was performed in order to analyse

the best pdfs to be used for random variables gener

ation. Nevertheless, Kolmolgorov-Smirnov revealed that normality could not be rejected for a level of confidence of 95%, for the waves' height and current velocity. Water depth wasn't tested due to the low amount of variance present in the original database. It would be important to analyse this aspect for other distributions, namely, the commonly used Rayleigh distribution for the waves' height. Including the hypothesis testing with more robust methods such as Shapiro Wilk test. This was not the main objective of the present study and therefore table 1 was considered for further application. 2.3 Failure Criteria, Performance Function, Safety Factor In order to obtain the probability of failure, the socalled performance function (g) must be considered. The g function expresses the system's behaviour in terms of its safety limit and the collapse situation. In scientific terms g can also be called the safety margin (M): Equation (2) translates to the following failure criteria: • If the critical shear stress (τ crres – obtained from the Shields parameter), that defines the threshold of motion for the top layer stones, is lower than the predicted critical shear stress due to marine environment (τ crpred) defined in equation (1), as in from De Vos (2011), the protection blocks will become unstable and the protection fails. For the top layer of the protection, the value of τ crres is computed at D 67.5 with the input parameters presented in section 3, which correspond to the same ones used in the previous date base. To compute τ crres the following equation can be used: where s = ρ s /ρ w is the soil to water density ratio, g is the gravitational acceleration, θ_{cr}, is the critical Shields parameter, defined in the original database as 0.035. The value of $D_{67.5}$ is obtained through the logarithmic relation of the sediments'size expressed in De Vos (2011) which depends on D_{50}. Based on the safety margin concept the following safety factor (SF) can be defined: The structure fails if SF < 1, since this means that the predicted shear stress produced by waves and currents exceeded the shear stress for threshold of motion in the top layer. The same interpretation for the performance function can be made, although for the limit g < 0. Note that SF depends on D_{50}, both through equations (1) and (3), meaning that once SF is defined a value of D_{50} is obtainable.

Table 2. Input parameters of the application case for a rip rap scour protection design, monopile diameter of 5 meters.

Input Parameter	μ	s
Water depth (d)	20 m	0.5 m
Current velocity (U_c)	1.5 m/s	0.7 m/s
Significant wave height (H_s)	6.5 m	0.5 m
Stones density (ρ_s)	2650 kg/m^3	0 kg/m^3
Water density (ρ_w)	1025 kg/m^3	0 kg/m^3
Shields Parameter (θ)	0.035	0
Sediment Uniformity Parameter ($\sigma = D_{85}/D_{15}$)	2.5	0

2.4 Probabilities of failure and Convergence

The probabilities of failure were computed using Monte Carlo Simulations according equation (2) and applying equation (5). N corresponds to the number of simulations performed. Independence between vari ables was assumed, as discussed in section 3, yielding a conservative estimate of P_f.

Probabilistic convergence was achieved by the Latin Hypercube Algorithm (Shields & Zhang, 2016).

Results showed a stabilization of P_f lower than 0.02%.

3 RESULTS AND DISCUSSION

3.1 Probabilities of failure and Safety factors

The present study obtained the probabilities of failure associated to each value of SF, according several values of the mean diameter of the rip-rap stones (D_{50}).Aiming at DeVos (2011) prototype scale fitting, equation (1), the following inputs were considered to assess the shear stress of currents and waves combined (τ_{crpred}). The μ and s respectively stand for the mean and standard deviation, applied in Monte-Carlo simulations.

Note that s = 0 means that the variable was considered to be constant. The values were defined as in De Vos (2011) application example and are shown in table 2.

As in the upper mentioned research, the relationship between the waves' period (T_s) and H_s was retained as $H_s = 4.4(T_s)^{0.5}$, while the waves'length L was obtained as Nielsen (1982).

In order to analyse the expected diameters of the block, several values of D_{50} were computed to obtain their association to each safety factor (SF), as if this relation could be considered deterministic.

However, since the variables considered are

stochastic in nature, it is important to perform the

analysis of the probability of failure according to SF.

The variability of each input variable can lead to

failure, even in a situation where a SF was defined to

ensure the security of the protection. Table 3. Probability of failure Pf (SF < SFi) vs. D50. SF (–) D 50 (m) P f [0;1] 0.80 0.26 0.98 0.90 0.32 0.79 1.00 0.39 0.52 1.10 0.47 0.29 1.20 0.57 0.14 1.30 0.68 0.064 1.40 0.81 0.025 1.50 0.96 0.0088 In other terms, due to the variability of scour phenomena and hydrodynamic variables there is a possibility that the design of the protection is being made for $D_{50} = x_i$ (expecting a $SF = SF_i > 1$) and failure still occurs, because the real value of SF is $SF_i < 1$. According the failure criteria previously defined, the following probabilities of failure were obtained for each design pair (D_{50} ; SF). Table 3 expresses the probability of a certain SF being lower than 1, when the scour protection is designed according a certain stone size and safety factor: Assuming a safety factor of SF = 1.1, which corresponds to $D_{50} = 0.47$ m, it would be expectable that the resistant shear stress, τ_{crres} , would overcome the critical shear stress produced by the local waves and currents, i.e. acting shear stress τ_{crpred} , by 10%. In a deterministic perspective this could provide a false sense of security. In fact, for the application case hereby assumed, P_f reaches 29%, which is quite high, if we consider the financial investments made in these structures installation and associated offshore operations. Note that similarly to scour phenomena in bridge piers (Johnson, 1992) the scour in offshore monopiles also presents about 50% chance of failure for SF = 1, which is the typical case of loads matching the resistances. In case of designing the scour protection for SF = 1.2 the required mean diameter would be $D_{50} = 0.57$ m. However it could be noted that the variability assumed for H_s , U_c , L, T_s leads to an actual SF < 1 in 14% of the times, meaning that the protection fails, increasing the instability of the monopile foundation. To achieve a probability of failure with an order of magnitude around 10^{-3} the required safety factor should be above 1.5. In the cases where SF < 1, the protection presents a very high rate of failure, which is understandable, since the design is being made with a very small dimension of the blocks, meaning that they do not present a resistant shear stress that can support the sea state conditions. No calculations were made for smaller D_{50} values than the ones associated to SF = 0.8, since this case is already deep into failure zone. Note that this safety factor already presents P_f = 98%, which is clearly

Figure 2. Safety Factors and Probabilities of failure; Equation applicable to the range of factors considered and the application case previously defined.

an insufficient scour protection design, for a typical monopile foundation.

Another interesting aspect is that SF ranging from 1.2 to 1.5 leads to dimensions of the blocks that might be too large for fall pipe installation made through vessels. A value of D_{50} = 0.96 m is clearly the case where P_f is low (0.008 = 0.88%) but the blocks installation might not be feasible, in offshore operations.

This points out to the required balance between the acceptable value of P_f (and SF) and the suitable dimension of the units used in the protection layer of the rip-rap system. Nevertheless, the present approach enables the calculation of a measure of risk that can be associated to the typical safety factor concept.

It is obvious that the safety factor concept, i.e. the failure criteria adopted, has a clear influence on the probabilities obtained. If the failure criteria is too restrict, the probabilities might increase and this will depend on a suitable criteria choice.

The possibility of using a more “loose” criteria is being studied by the authors in order to assess the probabilities of failure for dynamic scour protec tions, where a certain amount of movement is allowed to the top layer stones. For this dynamic concept it would be possible to decrease the blocks’ dimensions, for the same order of magnitude of P_f, reducing the costs of sediment material, installation and maritime transportation.

The following figure provides the outputs shown in the previous table, with P_f in a logarithmic scale:

Due to the previously mentioned relation between SF and D_{50} no calculations were made for safety fac tors above 1.5, since this would lead to non-practical dimensions for the units in the protection layer.

It is important to stress that the probabilities obtained remain valid for the application case pre viously defined. Based on Muzzammil & Siddiqui (2009) and Chang (1994) it is expectable that the out puts from figure 2 might be slightly overestimated.

This could be mainly due to two facts: Figure 3. Relation between safety factor and blocks mean diameter. • Normal Distributions were assumed for the random variables generation; • Independent variables were considered, i.e. correlation was neglected. According to Chang (1994) both assumptions yield conservative estimations of P_f. Nevertheless, this work could be improved by a deeper study concerning the adequacy of the probability distribution functions used in each variable and also by analysing the influence of correlation in the probability of failure. These facts should be considered in further research. Safety factors yield an intuitive interpretation for global safety. However as table 3 and figure 2 show, they offer no estimation of the risk involved in their use for design purposes. Applying a fit equation between the safety factors and the probabilities of failure, shown in figure 2, it is possible to find an equation, with an R^2 coefficient slightly above of 0.96, in the range of SF = [0.8;1.5]. The equation presented is defined as follows: By analytically associating SF and Pf it becomes possible to design the blocks dimensions, according to a predefined value of the probability of failure. A brief application of this concept is presented in the following section. Although the previous equation aims to be a best fit model for the considered range of SFs presented, it provides a slight overestimation for SF ≥ 1.4. For the values of safety factor bellow this limit, an underestimation is obtained. A possible way to account for this fact can be deducted from the previous figure and applied to design methodologies. Figure 3 shows the increasing stability of the protection, with the increasing D_{50}.

The simplest way to account for the differences in

the proposed fitting is to round up the final value of

D_{50} for the predefined P_f. If the protection stability

increases with the increasing mean dimension of the

blocks, the round up procedure will ensure that a con

servative perspective is being yield during the design

process.

In De Vos (2011) for a similar application case,

without considering the variability in the input param

eters, a statically stable design is proposed with a

D_{50} = 0.496 m.

By applying the present methodology, with the

distributions and variabilities already mentioned, it is possible to find an associated SF = 1.125, which relates to P_f = 17.1%.

The lifetime of offshore wind turbines is usually defined for a minimum of 20-25 years (Bhattacharya, 2014). Therefore, it is crucial to quantify the uncertainty of the sea state conditions (H_s, U_c, L, T_s) near the offshore structure. In this example, the protection of the foundation, designed for D_{50} = 0.496 m, could fail if a worst case scenario occurs, for the conditions defined in table 2, which might occur within the considered lifetime period.

On the other side, a suitable balance between SF and D_{50} should be reached in order to avoid bad cost/benefit ratios, when designing and operating in offshore conditions, where the foundation costs can easily reach 25-34% of the total investment made (Bhattacharya, 2014).

3.2 Example of Application

Assuming the same case study presented in section 3.1, this methodology could be applied in order to design the scour protection with a predefined value of P_f. The standard procedure would consist in the following steps:

- Define a value of acceptable P_f taking into consideration the amount of uncertainty of sea state conditions, for the installation place;

- Use equation (6) to obtain SF ;
- Use equation (4) to calculate the minimum required D_{50} ;
- Round up the obtained value of D_{50} to nearest commercial size available;
- Analyse if the obtained D_{50} is suitable for pipe vessels installation, offshore operations and desired cost/benefit ratios.

If the last step proves to be unfeasible the procedure should be repeated, although assuming that a higher risk is being taken in the investment, since the P_f will increase. In order to maximize the reliability of the system and minimize resources consumption, the designer and contractor companies should also assess which option will lead to lower costs:

- If it is a slightly larger value of the protection blocks and a lower value of the failure probability;
- Or if the dimensions of the blocks should be

Chang, C., Tung,Y.,Yang, J. (1994). Monte carlo simulation for correlated variables with marginal distributions. J. Hydraulic. Eng., vol 120(3), 313-331.

Cremona, C. 2011. Structural performance – Probability based assessment. Ed. JohnWiley & Sons Inc., ISBN:978 1-84821-236-7.

Das, P., Zhang, W. 2003. Theoretical methods of struc tural reliability. In: Guidance on structural reliability analysis of marine structures, pp.22–51. Department of Naval Architecture and Marine Engineering, Uni versity of Glasgow, University of Strathclyde, United Kingdom.

De Vos L., Rouck J., Troch P., Frigaard P. 2011. Empirical design of scour protections around monopile foundations. Part 1: Static approach, Coastal Engineering, vol 58, pp 540–553.

EWEA – European Wind Energy Association 2015. The European offshore wind industry – Key trends and statistics 1st half 2015. Report available at: http://www.ewea.org/statistics/offshore-statistics/. Accessed at 30th of September 2015.

Johnson, P., 1992. Reliability-Based Pier Scour Engineering. Journal of Hydraulic Engineering, vol. 118 (10), pp. 1344 1358, American Society of Civil Engineers, ISSN 0733

9429/92/0010-1344. LeBlanc, C. 2009. Design of Offshore Wind Turbine Support Structures: Selected topics in the field of geotechnical engineering. Department of Civil Engineering, Aalborg University. Muzzamil, M., Siddiqui, N., 2009.A reliability-based assessment of bridge pier scour in non-uniform sediments. Journal of Hydraulic Research, Vol. 47 (3), pp. 372–380, DOI:10.1080/00221686.2009.9522008. Nielsen, P. (1982) Explicit Formulae for Practical Wave Calculations, vol 6. n ◦ 4, pp.389–398. Prendergast, L., Gavin, K., Doherty, P. 2015. An investigation into the effect of scour on the natural frequency of an offshore wind turbine. Ocean Engineering, Vol. 101, pp. 1-11. DOI:

10.1016/j.oceaneng.2015.04.017. Shields, D., Zhang, J.(2016). The generalization of Latin hypercube sampling. Reliability Engineering and System Safety, 148, 96–108.
Sørensen, S., Ibsen, L. 2013. Assessment of foundation design for offshore monopiles unprotected against scour. Ocean Engineering, vol. 63, pp. 17–25, http://dx.doi.org/10.1016/j.oceaneng.2013.01.016.
Whitehouse, R., Harris, J., Sutherland, J., Rees, J. 2011. The nature of scour development and scour protection at offshore windfarm foundations. Marine Pollution Bulletin, Vol. 62, pp.73-88, DOI: 10.1016/j.marpolbul.2010.09.007.
Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Failure of fluvial dikes: how does the flow in the main channel influence the breach development?

I. Rifai

ArGEnCo Department, Research Group Hydraulics in Environmental and Civil Engineering (HECE),

University of Liège, Liège, Belgium

EDF R&D, National Laboratory for Hydraulics and Environment, Chatou, France

Saint Venant Laboratory for Hydraulics, Chatou, France

S. Erpicum & P. Archambeau

ArGEnCo Department, Research Group Hydraulics in Environmental and Civil Engineering (HECE),

University of Liège, Liège, Belgium

D. Violeau

EDF R&D, National Laboratory for Hydraulics and Environment, Chatou, France

Saint Venant Laboratory for Hydraulics, Chatou, France

M. Pirotton & B. Dewals

ArGEnCo Department, Research Group Hydraulics in Environmental and Civil Engineering (HECE), University of Liège, Liège, Belgium

K. El kadi Abderrezzak

EDF R&D, National Laboratory for Hydraulics and Environment, Chatou, France

Saint Venant Laboratory for Hydraulics, Chatou, France

ABSTRACT

Fluvial dikes are commonly constructed for flow channelization

and as flood defences; but in case of failure they lead to signifi

cant casualties and damages. Statistics show that overtopping is

the first cause of failure (Fry et al. 2012; Foster et al. 2000). Many

experimental studies on dike failure were conducted, but most of

them focused on normal configurations (i.e. where the flow is

perpendicular to the dike axis, such as in a dam break configu

ration), without accounting for the influence of the flow parallel

to the dike in the main channel. The aim of the present work is to

improve the current understanding of physical processes under

pinning gradual breaching of fluvial dikes by overtopping, taking

into account the flow parallel to the dike axis.

One challenging aspect of laboratory experiments of dike fail

ure is the continuous monitoring of the breach evolution. We have

developed a non-intrusive and distributed measurement tech

nique based on laser profilometry (Figure 1). The approach offers

a high degree of flexibility, since neither the location of the camera

nor that of the laser need to be known a priori (Rifai et al. 2016).

Results of the experimental tests unveil breach evolution

mechanisms that strongly differ from those occurring in a frontal

dike failure configuration. The breach expands almost only

towards downstream of the main channel. This expansion is

Figure 1. Main steps of the continuous 3D reconstruction

of the breach geometry by the laser profilometry: (a) images

of the experimental dike failure and processing, (b) final

Continuous grid monitoring to support sediment management techniques

T. Van Hoestenberghe & R. Vanthillo

Fluves, Ghent, Belgium

P. Heidinger & J. Dornstädter

GTC Kappelmeyer, Germany

N. Dezillie & N. Van Ransbeeck

VMM, Brussels, Belgium

ABSTRACT: Fluves monitors since November 2015 continuously a sediment trap with dimensions 200 × 20 m

managed by the Flemish Environmental Agency (VMM) in Belgium. The continuous follow-up of sedimentation

of the trap must provide insights on temporal and spatial evolution of trapping efficiency.VMM needs the insights

for operational decisions when to dredge the trap and for optimizing the design of future traps. The installed

measuring system is based on distributed temperature sensing with a fiber optic cable of more than 2 km. A

digital terrain model (DTM) of the sediment trap is transmitted to the client on a hourly to daily basis. Results

after winter floods from November 2015 till February 2016 show a significant spatial variation in sedimentation

through the sedimentation trap. Also, zones with different temporal evolution of filling of the trap could be

observed. The technique shows great potential for detailed spatial and temporal observation of sedimentation

processes in large areas.

1 PROJECT GOAL

VMM, responsible for unnavigable watercourses in

Flanders, wants to have more insights in sediment

dynamics in the sediment traps installed on their water

courses. The annual cost of dredging is high, and sedi

ment traps installed in rivers or waterways can decrease

dredging costs by concentrating sedimentation, such

as on the Elbe estuary near Hamburg (HPA, 2013).

In Flanders many sediment traps are recently

installed in the upstream area's to trap sediment before

contamination occurs more downstream (Ferket et al.,

2014). Sediment traps are nowadays roughly mon

itored by bathymetric measurements. The trapping efficiency, which describes the fraction of total sed iment entering the trap that remains in the trap, is in most cases unknown.

By monitoring the sedimentation continuously, it becomes clear when the trapping efficiency start to decrease. With this knowledge, decision can be taken when to dredge the trap. But more important, these insights will enable the river manager to enhance the design for future traps. Also, the measurement data can be very useful for erosion and sediment transport modeling.

2 MATERIALS & METHODS

2.1 Distributed temperature sensing

Fiber-optic Distributed Temperature Sensing (DTS) has become a popular tool in environmental monitoring. With DTS, fiber-optic cables are used as temperature sensors. Temperature can be measured at resolutions of 25 cm to 1 m along cables several kilometers in length, and the measurement interval can be under a minute. It has been used to provide both qualitative and quantitative information on many processes with a thermal signature (Solcerova et al., 2015). Two types of DTS measurements are used. In “Passive DTS”, cables are used to monitor the natural temperature dynamics in the soil. Soil thermal properties can be calculated from the difference in amplitude and phase between temperature measurements at different depths. In “Active DTS”, the cable is heated, and the fiber monitors temperature changes in the cable. The temperature response in the fiber can be directly related to the soil moisture, or equivalently the thermal properties of the surrounding soil. In addition to monitoring soil temperature and moisture, the thermal conductivity determined from either Passive or Active DTS can be combined with the temperature profile to estimate the soil heat flux (Steele-Dunne et al., 2012). Sebok et al. (2012) conducted measurements with a 750 meters fiber optic cable and DTS in the Holtum stream in Western Denmark to find spatial and temporal evolution of newly deposited sediments. They could identify the existence of scouring, migration, and re-deposition events. For this project, a fiber optic cable of more than 2 km was installed in a grid pattern in a sediment trap. Spatial resolution of the measurement grid is almost every m 2 . From the temperature measurements, the sediment height on the cable could be quantified.

Figure 1. Position of the sediment trap in upstream area of

Molenbeek Hasselt (Flanders, Belgium).

2.2 Site characteristics

In figure 1, the location of the sediment trap is shown.

The trap is situated in the southern part of Flanders,

near Geraardsbergen.

The trap is installed in a flood control area (FCA)

on the river ‘Moenebroekbeek’, an upstream branch of

the Dender river, a tributary of the Scheldt river. Aver

age river discharge is 500 l/s. The sediment type that

enters the FCA is fine loam, typically with medium

grain size of 10–20 μm. The sediment transported by

the river to the FCA and deposited in the trap has

mainly it’s origin in soil erosion processes from the

adjacent agricultural land. The estimated annual sed

iment transport towards the trap is 1500 ton. With an

expected trapping efficiency of 25%, the trap would

need to be dredged each 10 years.

The sediment trap is an enlargement and deepening

of the river. The width is on average 20 m and the total length is 200 m. The bottom of the trap is at level 23 m TAW (TweedeAlgemeneWaterpassing = local sea reference level), somewhat lower than the incoming river. A gate at the downstream part of the FCA controls the filling of the FCA.

2.3 Measuring method

The sediment height is measured throughout the pool by combining direct and indirect measurements. At 20 locations, evenly distributed in the pool, the soil/water interface is measured directly by winding the fiber optic cable around vertical poles (Figure 2). Figure 2. Sediment trap after installation of the monitoring system with 20 vertical poles throughout the trap. In order to determine the difference in thermal parameters of the water and sediment, the “Active DTS” – method is used. An electrical current is sent through an electrical conductor, installed together with the fibre optic cable. This cable system can be heated up considerably. The temperature increase of the surroundings of the cable is dependent on the heat capacity and heat conductivity of the water or sediment. Water, especially when flowing, cause a reduced heat up of the cable compared to sediment (Dornstädter, 2001). The DTS unit measures the thermal response of the heating for all measurement points along the cable, one value every 25 cm. The DTS installed for this application is from manufacturar APSensing (Figure 3). GTC Kappelmeyer GmbH assisted Fluves for the installation of this system. When the cable is surrounded by water, the thermal response will be high. When the cable is buried in sediments, the response will be low. Because the installation layout is known, the results can be transformed from fiber meter distance to height of the poles. Then the threshold from low to high thermal responses marks the upper boundary of sedimentation. Sediment height can be measured with an accuracy of 1 cm at these 20 poles. In between the poles, the cable is installed horizontally on top of the base level of the sediment trap. To be able to measure the sediment accumulation indirectely in between the poles, the hybrid fiber optic cable is configured in a pattern as described in Figure 4. By using a long heat pulse (multiple hours), the thermal response can be analysed in areas at further distance from the cable. The heating curve will be influenced by the thickness of the sediment layer and the thermal diffusivity. The thermal diffusivity is an unknown parameter, dependent amongst others on soil moisture content and density of the sediment. But at the poles, where the thickness of the sediment is known, the thermal diffusivity can be calculated. So the poles measurement serve as a (continuous) calibration for the measurements in between the poles. The accuracy of these indirect sediment height measurements in between the cables is expected to be +-10 cm. The cable was installed in one deployment during October 2015, combining direct and indirect measurement installation.The system became fully operational by the end of October 2015.

Figure 3. DTS instrument from APSensing and auxiliary

equipment for the heating cycles. The system is remotely

controlled.

Figure 4. Sediment trap with vertical poles (red) and hori

zontal layout of fiber cable throughout the trap, 1 week after

installation. At each pole, the sedimentation height is indi

cated (in cm). Blue color indicates no coverage of the cable

with sediment, yellow indicates a covered cable.The blue line

indicates the flow direction.

3 RESULTS

3.1 Spatial variation

In Figure 5, the results are shown for the direct mea

surements after several flooding events in the period

November 2015 till beginning of February 2016.

During three flood events, the FCA was filled. Fill

ing started in 2015 at an upstream water level of

24.25 mTAW and in 2016 at an upstream water level

of 25.00 mTAW. During the flood event of 20/11/2015,

the FCA was filled for 15 h. During the flood events

from 15 till 16 January 2016 and 9 till 11 February

2016, the FCA was filled for 16 h respectively 24 h,

but filling started only at 25 mTAW. There were sev

eral minor flood events where the water level upstream
Figure 5. Sedimentation height (in cm) between begin February 2016 and November 2015 at the 20 direct measurement locations. Red and brown zones indicate the locations with respectively less and more sedimentation than the average sedimentation rate (i.e. 5 cm). the gate did not reach 25 mTAW. The flood event from 9 till 11 February 2016 was the event with most flood volume and longest retention period of flood water in the FCA. The sedimentation height between begin February 2016 and November 2015 is on average 5 cm. This is less than what was expected before the start of the measurements. At the most upstream area, significantly less sedimentation is measured (less than 3 cm). In contrast, in the middle of the trap, a zone with distinctly more sedimentation can be observed, up to 10 cm in the middle. In the most downstream part of the pond, a uniform sedimentation of 4 to 5 cm is observed. A reason for these spatial variation could not yet be identified. There is no correlation between initial base level of a specific location in the pond and the corresponding sedimentation rate. Local eddies and local vegetal obstruction are the most plausible explanation of these spatial varation. No results are given for the indirect measurements between the poles. The total sedimentation layer is too low for a statistically significant difference between the different locations along the cable, since the accuracy of the indirect system is (a sedimentation height of) 10 cm. Once the sedimentation layer has increased above 10 cm, data processing will lead to estimated sediment heights along the cable. 3.2 Temporal variation Because sedimentation can be measured with an accuracy of 1 cm, sedimentation, erosion and redeposition can be monitored with great detail. In Figure 6, the evolution of sedimentation for the 4 poles with more than 7 cm sedimentation between November 2015 and February 2016 is shown. There is a correlation between sedimentation rate with the water height, but the

height and volume of flooding of the FCA is not the only explaining factor. The highest increase in sedimentation rate can be observed for a modest flood event on the 4th of January 2016. At this event, the FCA was not even filled since filling starts at a water level of 25 mTAW. One would expect much higher sedimentation rates

Figure 6. Time series of evolution of sedimentation for 4 poles with more than 7 cm sedimentation between November 2015 and February 2016. The blue curve is the water height (right Y-axis). The numbers 10, 11, 13 and 18 refer to the location of the poles, as shown in Figure 3.

Figure 7. Time series of evolution of sedimentation for some poles between the flood event between 7 and 9 February 2016. The blue curve is the water height (rightY-axis). The numbers 5, 7, 12, 13 and 18 refer to the location of the poles, as shown in Figure 3.

for the higher flood events of 15–16 January and 8–10 February 2016, when the FCA was filled and current velocity slowed down.

Other observations are that at some locations there is some erosion directly after a flood event (decrease of sedimentation level), but this erosion is very limited. Once the sediment is deposited, it remains mostly stable. This also indicates that there are no significant consolidation processes, i.e. decrease of volume while mass conservation, once the sediment deposited in the trap.

In Figure 7, the evolution of sedimentation for

some poles between the flood event between 7 and

9 February 2016 is given. For this event, we see a slow

and modest sedimentation while the FCA is flooded.

For most locations, the maximum sedimentation

(only 2 to 3 cm) decreases a little (1 cm) when the FCA

empties.

4 CONCLUSIONS

With the recent developments in fiber optic measure

River flow analysis with adjoints – An efficient, universal methodology to

quantify spatial interactions and sensitivities

U.H. Merkel

UHM River Engineering, Karlsruhe, Germany

J. Riehme & U. Naumann

STCE, RWTH Aachen University, Germany

ABSTRACT: The TELEMAC-SUITE with Algorithmic Differentiation (TELEMAC-AD) quantifies the

impact of very high numbers of spatially distributed parameters on flow related problems in one step. This

technique is a revolutionary step forward in open channel flow simulations, as it solves previously unsolv

able or computationally very expensive problems. Classic simulation methods return the combined impact of

many parameters, whereas the adjoint version of TELEMAC-2D returns the individual influence of every single

parameter in one run. A wide range of new applications is now possible on basis of well validated source code:

Automatic optimization and calibration of flow relevant shapes, data assimilation, high resolution sensitivity

analysis or deciphering of superimposing flow effects.

1 INTRODUCTION

This article describes a new method for open chan

nel flow software which solves inverse flow problems.

Instead of traditional forward questions like

“Put water in here, how will it be distributed

within our project area?”

we solve INVERSE questions of the type

“We want specific flow related project result,

where and how do we have to modify our

river?”.

See Figure 1 for a simple example comparison:

– Fig. 1 (left) shows the traditional forward project:

Few input parameters that influence many target

parameters.

– Fig. 1 (middle) shows a typical inverse project:

Many input parameters have influence on one target

parameter.

– Fig. 1 (right) shows the solution of the inverse prob

lem through backward interpretation of forward

calculations.

Input parameters are any specific parameter in very

large numbers of geometric points (e.g. spatial variable

parameters like bed roughness or bathymetry), but also scalar parameters (e.g. discharge or an empiric formu

las coefficient). Each floating point value of a finite

elements or volumes program has a potential individ

ual influence. We want to quantify for all parameters

their relevance for the single resulting value, which

is the outflow discharge Q in fig. 1. The relevance is

described by the gradient of the output Q with respect to the inputs, which is the vector of partial derivatives ∂Q/∂ Input (i). If the hydraulic problem is based on a large number of influence factors, then the adjoint model of the TELEMAC-SUITE is the most efficient solver: The complete gradient can be computed by one run of the adjoint model. This adjoint version, called TELEMAC-AD, was generated by the authors with the help of the differentiation-enabled NAG FORTRAN compiler. This compiler is a joint development of the institute STCE at RWTH Aachen University, Germany, NAG, UK, and the University of Hertfordshire, UK. After a short introduction (Chapter 1) few examples illustrate the potential of TELEMAC-AD (Chapter 2). A basic introduction of theAD concept follows in (Chapter 3). More detailed information on the functionality of TELEMAC can be found in [Villaret, 2013] and [Kopmann, 2013]. 1.1 New applications The adjoint version of TELEMAC opens a door to a new range of hydraulic modeling applications. Typical applications are the impact quantification for any point of a TELEMAC-2D-3D or SISYPHE model. Examples: Any node I (and its connected parameters velocity V, water level W.L., manning's n . . .) can – influence the flow conditions at a power plant intake and the maximum energy level which results in more or less electrical energy. – influence robustness and sensitivity of hydraulic relevant structures and hydraulic driven processes of all kinds (like dams, bridges, flood protection

Figure 1. Left: Common forward problem: Few parameters influence many, dependencies easy to calculate. Middle: Inverse

problem. Few parameters depend on many, sometimes millions of geometry points. Right: The adjoint method solves until

now unsolvable inverse problems by interpreting the forward problem backward and producing up to millions of independent

sensitivities in one work step.

measures, morphological aspects, a.o.) [Kopmann, 2012].

In combination with gradient based optimization methods the adjoint technology can be used for

- automatic calibration of thousands of roughness val ues at the same time, e.g. to fit water levels to measured values (until today an unsolved every day problem).

- semiautomatic modification of single point coor dinates to modify flow conditions according to a target function.Airfoils and drag coefficients of cars have already been optimized withAD in mechanical engineering (shape optimization).

1.2 Using the classic forward calculation

The influence of boundary & initial conditions like W .L. and V (i), i = 1, . . . , N , on a target parameter like Q for questions as in Fig. 1 (middle) is described with the gradient (left part of formula 1):

Until today the differential analysis is the common way to analyze and optimize the flow field or mor phology (right part of formula 1). Every element of the gradient approximation has to be computed separately with slightly different input parameters. For millions of points millions of calculations are necessary or the points have to be grouped, what will bias the results. Since the early days of physical flow models the real target is to minimize the difference between a real and a desired flow field (maximum erosion, water levels, velocity field etc.). The main question in many cases is that one few parameter (the difference) depends on many.

This can be any mathematically relevant variable, but especially for open channel flow it is the spa tial distribution of parameters in big FE meshes. But an analysis for millions of points (each one is a bathymetric feature) is too expensive normally. 1.3 Backward interpretation with adjoint models The adjoint model of TELEMAC-AD computes the entire gradient from formula 1 in only 1 forward run and its following backward interpretation. For this, TELEMAC-AD records every relevant instruction during a forward evaluation in the so called “tape”. At the end of the forward run all target values and the process flow are stored in the tape. The backward interpretation propagates the adjoints (derivatives) from the target parameters (at the end of the tape) to the input parameters (at the beginning of the tape). During this process every single instruction is interpreted by its adjoint version. The adjoints obtained for the input parameters can now be used as dependency information, for robustness or sensitivity analyses or in a next step for gradient based optimization methods. 2 EXAMPLES Three examples illustrate the potential of adjoints. They are based on the open source version of TELEMAC. – A small scale design problem, which is dominated by unclear effects of various obstacles (bathymetry elements) on both flood safety and maximum velocities. – The diffusion of a tracer after a confluence, based on the shape of the confluence. – A hydraulic 2DTELEMAC river model is examined to quantify the influence of 95000 spatially distributed parameters on the shear stress in 1 specific point. They include sediment transport, tracer distribution, automatic calibration, optimization strategies and uncertainty quantifications. First 3rd party users published their own experiences with adjoints and sensitivities produced by TELEMAC-AD: – Villaret et al. (2015) from HR-Wallingford/EDF used the sensitivities of TELEMAC-AD and Sisyphe-AD for a First Order Second Moment

Figure 2. Urban river channel with several design limitations: The rock obstacles for a future slalom kayak training stretch

shall not produce to fast local velocities and are not allowed to decrease the free board. Middle: flow conditions at 50 cms.

Left: Change of the peak velocity in Zone A in dependency of the surrounding bathymetry (obstacles). Right: Dependency of

the minimum freeboard at any place on the obstacles bathymetry at 400 cms (100 year flood event).

Method (FOSM) based uncertainty analysis in sed

iment transport models.

– Trung Hieu Mai (2015) did a FOSM analysis at a

River Rhine stretch and compared it with a Monte

Carlo Simulation.

– Monika Schäfer (2013) used the adjoints of

TELEMAC-AD for an automatic calibration of a

flume experiment. She compared the most common

gradient based linear optimization algorithms for

usage with TELEMAC-AD results.

2.1 Example 1: Analysis of obstacles on maximum velocities in a certain point and on flood safety

Figure 2 shows an inner city river section which has

to be ecologically and optically improved. Many dif ferent interest groups force the designing engineer to fine tune the bathymetry: The city sports department wants obstacles for a slalom kayak training course, the environmental experts want the peak velocities to be below a threshold for a better fish migration. The disaster management needs sufficient free board. And everything is connected to the position and size of the obstacles and the bathymetry in general. Two typical inverse questions arise and both are based on one and the same common forward calculation:

– The mean peak velocity U found in Zone A (marked in Fig. 2) is depended on potentially any surrounding geometry information in the 128 000 model points.

This results in a cost function of type – The minimum free board (in any boundary point) at a 100 year flood event is potentially depended on any surrounding geometry information in the 128 000 model points. This example demonstrates, that a cost function is potentially any post processing calculation that reduces the problem to one parameter and one forward run might be used for many different questions (=cost functions). In both cases the adjoint model computes the gradient of the 1-dimensional J with respect to the 128000-dimensional bottom elevation Z j . The visual interpretation of the adjoints (Fig. 2) shows exactly where a modification of the geometries helps most. (Dark red and blue zones.) An especially interesting aspect is the fact that the adjoints show a dependency to the steepness of the groyne like obstacles in Fig. 2, left. It is visible through the blue bottom halo around the affected obstacles red high nodes. Calculation time for the two cost functions and their adjoints is 137 min on a mobile Intel i7 CPU with 3.0 GHz (single core). Figure 3 is a simple application of this information: The adjoint map clearly shows an effect on the peak velocity for the quite distant groynes on the river’s left

Figure 3. The adjoints can be used as design guide for a

new bathymetry. Even the indirect influence of distant obstacles is

quantified. It would not be obvious to a designing engineer, that the selected groynes elevation has a significant influence on

the maximum velocity in Zone A. But the validation run with removed groyne shows a reduction of the peak velocity by 9%.

This is mainly an effect of more water staying on the river left side.

side. For a manual optimization the most distant one

is removed. The recalculation of the full model results

in a 9% reduction of the peak velocity.

A next step for the usage of the adjoints is the so

called “shape optimization”. The automatic manipula

tion of the bathymetry shall achieve the project’s target

by reducing the cost function.

An application of this automatic optimization tech

nique with adjoints is shown for sediment transport in

Merkel et al. (2013).

2.2 Example 2: Tracers and shape optimization

The academic example of a confluence zone in Fig. 4

illustrates the application possibility of TELEMAC

AD to tracer and shape optimization projects.

The tracer could be a concentration of a solved

chemical or a temperature.

The sharp edge right behind the confluence obvi

ously produces an eddy which prevents the current to attach immediately to the left wall. Therefore the left channels tracer plume penetrates deeper in the right channels clear water. The cost function J in this example describes the width of the plume at the models exiting cross section.

The X,Y and Z components of every nodes position are examined for their influence on the cost function. The adjoints $\nabla X J$, $\nabla Y J$ and $\nabla Z J$ obviously recommend a shift of boundary points right behind the confluence edge.

2.3 Example 3: Dependency of shear stress in a groin head scour

“221_Donau” is a public validation example for

TELEMAC-2D and it is available for download at OPENTELEMAC.ORG. It is used here as large scale example to demonstrate how the surrounding bathymetry z(i) and the long range neighborhoods roughness distribution ks(i) influence flow conditions in a certain point of a real river. The shear stress τ is the target of this analysis (averaged over 12 nodes in the middle of the groin head scour, see markers (3) and (6) in Figure 5). This real world example has highly complex multidimensional flow conditions with islands, groins, wet/dry changes, and many other flow interacting features which encrypt dependencies for the engineer by overlapping. The spatial interaction is broken down to local values for every point with TELEMAC2D-AD, here with focus on the roughness and elevation values of the 47500 points in the 2D mesh. The sensitivities are independently quantified in total for 95000 parameters. This example is chosen to proof the usability for practical purposes and to show that even effects of points in many kilometers distance can be quantified with high accuracy. Furthermore it shows the advantage of adjoint models versus FD modeling. 2.3.1 Classic solution with finite differences (FD) Solving per point independent dependency analysis requires a base calculation and 2 * 47500 full TELEMAC-2D calculations with a slight modification of one input parameter. Only this will proof spatial independent input-output dependencies. See formula 5 and 6:

Figure 4. Confluence of two channels (flow from left to right). The left one brings a tracer (dimensionless, contour color),

which is mixed with the clear water of the right channel. The inverse question is: How does every points X,Y,Z position affect

the tracers plume width at the model outflow? The answer to the question is the field of arrows (dConcentration/dMesh(x,y))

that point in a potential node shifts direction. The new node position would reduce the plume width.

One calculation runs 5min on a up to date desktop

computer, this means 330 days for all calculations or

a very expensive outsourcing on a bigger cluster.

And the result is only valid for:

- one parameter set (discharge, bathymetry, rough

ness, hysteresis . . .)

- the simulated time span

- very limited extrapolation, due to the nonlinearity

in many sub models (see Fig. 6!).

Practical projects usually observe many variants,

optimization projects even more, which leads to very

expensive computational costs, making this tech

nology economically unusable for most small and

medium size projects.

2.3.2 Adjoint solution (AD)

TELEMAC-AD calculates the full forward run and backward interpretation in 678 min on the same desk top computer, and returns all 95000 adjoints for the shear stress τ . The gain of computational speed equals the usage of an approx. 1000-core cluster when using the classic FD method. Some results are displayed in Fig. 5. At the time of writing TELEMAC-2D-AD still has a lot of optimization potential for computational speed, which means that a further speedup is expected after completion of the ongoing developments.

2.3.3 Interpretation of the resulting adjoints

Adjoints computed by TELEMAC-2D-AD describe the change of the output shear stress τ as a linear rela tion of its specific input. Therefore extrapolations for other input parameter sets can only be done with great care. The influence of the bathymetry on τ in the scour at the simulated flow conditions is dominated by the obvious separation of the 2 arms around the upstream island. If the southern arm, which has low flow, is lifted, then more water is pushed to the main channel (marker "1" in fig. 5). The same happens if the surrounding of the groin and the opposite site ("2") of the cross section are elevated. A kind of funnel effect increases the shear stress. Decreasing the scour itself increases the shear stress as well ("3"). The perspective view from downstream ("4") reveals that some other groins have a high impact, as they influence with their back draft the water level in the examined area. The lower groins, which are smaller and shaded by the bigger ones, therefore don't influence the examination zone anymore. Again the reader shall be warned that only a change of few decimeters in any topographic feature might change the result totally. The general noise in the adjoints origins from bumps and holes in the bathymetry. Bathymetry changes that follow the direction of the adjoints will smoothen the main channel and therefore accelerate flow and increase the shear stress. On the contrary for bed roughness a clear tendency is visible: the smoother the bed along the main channel, the higher the shear stress at the groin head scour. If the roughness gets higher on the opposite shore ("5"), more water is pressed to the scour. Classic methods (numerical and physical) would have given a rough idea about this dependency, but for the first time a hydraulic model can exactly define the spatial limits of the relevant area. The scour itself has a different tendency ("6"): Increasing the roughness in the target point itself obviously increases the local shear stress.

Figure 5. Result of a single Telemac-2D-AD calculation. Top: Dependency of the shear stress in the scour (marked) to

neighboring bathymetry information. Bottom: Influence of the roughness on the shear stress. [Per m 2]. Values are only valid

for the current flow conditions and setup! (Modeled river section length: 7 km).

3 BACKGROUND KNOWLEDGE:

ALGORITHMIC DIFFERENTIATION

Algorithmic differentiation (AD) is a mathematical method that extends existing computer programs in a way that for a priori chosen results their dependency to a priori chosen input variables is additionally cal culated. AD tools work on the original source code, and not as a new implementation of the mathematic model. Differentiated models obtained by AD com pute derivatives in machine precision for a given input parameter set.

The main benefit is, that the calculation results of the forward run are binary identical as the source code stays untouched and the additional calculated analytically differentiated sensitivities are in machine precision. The upcoming fig. 6 shows the comparison to finite differences: The displayed example func tion returns the dependency, a tangent, of the water level W.L. versus roughness ks. Valid for only one examination ks setup.

AD analytically calculates the tangent to the func tion, while the commonly used finite differences only give an approximation to the tangent. FD's resulting approximation error depends on the ks step wide for the finite difference comparison. But if the FD step is chosen very small, the model might encounter instabil ities. FD therefore always ends up with trial and error for the step wide, normally with the result of a slightly larger than necessary step wide.

Since some time aerospace, mechanical engineer ing and meteorology are working successfully with adjoint models. They use AD for uncertainty quantifi cation, data assimilation, optimization strategies and inverse problems. Inverse problems in hydraulic engineering are for example the quantification of boundary and initial conditions for given results. The principles of AD are based on the fact, that every calculation is finally a combination of basic operations (+, −, ∗, /, exp, sin, . . .) with well known differentiation rules. All more complex formulas are a sequence of these; derivatives of basic operations are combined by the chain rule to derivatives of sequences. Two basic techniques are used for first order derivatives: – Tangent linear models (forward models) work the same direction as finite difference (FD) approximations of derivatives, but at machine precision. Fig. 6 shows this advantage against FD, which has to use 2 calculation results for each derivative. – Adjoint models (inverse interpretation of the forward model) propagate the adjoint (derivative) from the final results back to initial and boundary conditions. For the backward interpretation the full path of the forward calculation has to be recorded. If the forward model has just a single target parameter value, the vector of derivatives (the gradient) can be calculated in only one backward interpretation. Second or higher order derivatives can be calculated by combining the basic techniques. Higher order derivatives might speed up optimization processes significantly. For further information see [Griewank, 2008].TheAD-enabled NAG Fortran Compiler [NAG, 2013], developed at the Institute "Software and Tools for Computational Engineering" (STCE, University RWTHAachen) is a commercial extension of the NAG FORTRAN compiler (Numerical Algorithm Group,

Figure 6. Water level as a function of the roughness ks.

Normally the dependency is not linear. Dependency is

described as first derivation, this means the ascent of the tan

gent is defined in only one point for algorithmic (~ analytic)

differentiation and in two for finite differences.

Oxford, UK). It uses a hybrid technique of source code

transformation and an efficient overloading based run

time library. [Rauser et. al.] discuss this methods in

detail. The hybrid approach allows an efficient differ

entiation of large projects like the TELEMAC-SUITE.

For practical usage the recording of all itera

tions and all operations within these iterations is a

memory consumption problem. For the 221_Donau

example 10TB of RAM would be necessary. There

fore TELEMAC had to be extended by a so called

checkpointing technology. What means that every cal

culation can be saved to RAM and restarted from RAM with binary identically results at any point in time, with

only a minimum recording of variables.

For the backward propagation of the adjoints (back

ward time step wise) the necessary detail information

about subprogram internal operations is recalculated

from the checkpoints in reverse order. This increases

calculation effort by 200%, but reduces the RAM

usage, as the minimum checkpointing system only

needs 300MB for 1000 time steps in the 221_Donau

example, plus 10GB for the current time step.

4 CONCLUSIONS

The methods of reverse interpretation and algorith

mic differentiation enable a very fast quantification

Comparative use of FVM and integral approach for computation

of water flow in a coiled pipe and a surge tank

Muris Torlak & Berin Šeta [1]

University of Sarajevo – Mechanical Engineering Faculty,
Sarajevo, Bosnia-Herzegovina

[1] TENSOR Technology Doo, Branilaca Šipa, Sarajevo,
Bosnia-Herzegovina

Adis Bubalo

Elektroprivreda BiH – HE na Neretvi, Jablanica,
Bosnia-Herzegovina

ABSTRACT: This paper presents results of combined 3D
computational fluid dynamics (CFD) analysis and 1D

integral analysis of water flow in hydraulic systems. CFD
analysis is based on solution of Navier-Stokes equations

for incompressible flows using a finite-volume method (FVM), which requires discretization of space, time and governing equations, leading thus to a relatively large computational effort, but delivering detailed information on various flow properties. Anticipating faster computations, tailored methods for integral 1D analysis of two different problems considered in this paper are also applied. The examples presented include (i) FVM calculation of pressure drops and corresponding friction factors in a helically coiled pipe at different flow regimes, which are compared with the results of relations from literature implemented in the integral model for curved pipes, as well as (ii) the use of another integral model and FVM for calculation of water elevation in a surge tank for which measured data are available.

1 INTRODUCTION

Hydraulic problems arise in a wide variety of technical systems (for example: pipeline transport, hydropower plants, water supply and distribution, building services, heating/cooling systems etc.). Not rarely, their proper understanding and effective solution play a key role in development and optimization of individual parts or even of complete systems. Traditionally, scientific analysis, technological development and product improvement have been focused on experimental investigations and testing. However, due to ever shorter time-to-market demands as well as increasing needs for flexibility, which are strongly supported by the giant advances in hardware and software solutions, the use of computer simulations in development and optimization processes has become inevitable practice. Methods used in the field of computational fluid dynamics (CFD) have been proven in effective treatment of a number of complex physical fluid flow problems, also including hydraulic phenomena. On the other hand, these methods are not always in line with modern design procedures. Necessity for analysis of large numbers of variants, as well as permanent demands to keep the processing times considerably shorter and computer resource requirements limited, make CFD methods sometimes unfavorable. The reason for this is the underlying mathematical method based on discretization of the space and the time interval considered, yielding thus the number of equations to be solved related to the level of discretization details. Depending on the physical scales arising in the problem under consideration, the required levels of discretization in a number of cases might be practically unaffordable, for example when turbulence effects play significant role. Integral approach however offers the possibility to describe the problems with considerably less equations ignoring many detailed properties. The drawbacks of this approach are a limited number of resulting fluid flow properties, and the fact that different problems often require different, specifically tailored mathematical models and solution algorithms. In this paper, a comparison between CFD approach based on a finite-volume method and integral approach to solution of two selected hydraulic problems is presented: (a) pressure drop and friction factor assessment in helically coiled pipe and (b) water elevation in a surge tank of a hydropower plant.

2 SOLUTION OF NAVIER-STOKES EQUATIONS

Motion of a single-phase fluid in a control volume V, bounded by the closed control surface S, is described by the Navier-Stokes equations, here presented in integral form in which the flow properties represent local, instantaneous values (Ferziger & Perić 2013): where t is the time, ρ is the density, $\vec{v}$ is the velocity vector, and f_b is the vector of body forces taking the gravity effects into account. The stress tensor σ is related to the strain rate by the constitutive equation for incompressible fluids: $\sigma = -pI + 2\mu\dot{\varepsilon}$, where p is the pressure, I is the unit tensor, μ is the dynamic viscosity, and $\dot{\varepsilon}$ is the strain rate tensor.

Two-phase flow of water and air in hydraulic systems can often be regarded as flow of immiscible phases. The separating surface, interface between the phases usually referred to as liquid free surface, is clearly distinguished. A typical modeling practice in such a case is to follow volume-of-fluid approach with interface capturing (Muzaferija & Perić 1998, Muzaferija et al. 1998), observing the flow of an effective fluid mixture in a control volume under consideration. The two phases are assumed to share the velocity and the pressure. The volume-averaged properties of the individual phases, density and viscosity, are then regarded as physical properties of the effective fluid mixture. Distribution of the transported phase (typically, the liquid phase) is described by the scalar transport equation for the phase volume fraction c: Both water and air are in this study regarded as incompressible, hence density ρ is assumed to be constant in space and time, and may be omitted from Equations 1–2. Turbulence effects are captured by two additional transport equations for the turbulent kinetic energy and the specific dissipation rate following the two-layer, realizable k-ε turbulence model with treatment of all y + values (Shih et al. 1994, Rodi 1991) or k-ω SST turbulence model (Menter 1994).

2.1 Finite-volume method

The computational mesh is generated, accommodating the region of interest. The mesh consists of an unstructured set of contiguous cells bounded by a number of flat faces. Having adopted the cells as control volumes, Equations 1–3 and turbulent transport equations are discretized using a finite-volume method – FVM (Ferziger & Peric´ 2013, Demirdžic´ & Muzaferija 1995). The cell centroids are chosen to be computational points, at which all solution variables are calculated. The integrals are approximated using the midpoint rule. A linear-upwind scheme is used for discretization of the convective terms in Equations 1–2, while high-resolution interface capturing scheme HRIC (Muzaferija & Peric´ 1998, Muzaferija et al. 1998) is used for the corresponding term in Equation 3. The stress tensor at the control surface, arising in the viscous term of

Equation 2, contains the strain rate and herewith the velocity gradients which are approximated by differencing the variable values from the adjacent computational points. The additional gradients at the cell centers are calculated as a blend of their Gauss method approximation and the weighted least-square approximation. The blending is done in order to keep robust performance on mesh cells of different qualities.Additionally, appropriate limiting may be applied taking into account the variable distribution over the neighbor cells. In transient problems, the time interval is divided into a number of the time steps, usually of the same size. Implicit time integration is applied due its stability, typically with Euler or three-time-level scheme approximation of the time derivatives (Ferziger & Peric´ 2013). The resulting systems of equations are assembled for each solution variable separately. They are processed sequentially in turn until the prescribed tolerance level is reached, solving them by virtue of an algebraic multigrid method. All CFD calculations in this study are performed using the commercial software STAR-CCM+®. The details on the implementation of the numerical method can be found in its documentation (CD-adapco 2014). 3 INTEGRAL 1D APPROACH In this section, mathematical models and methods for their solution are briefly described. The both models are implemented in two in-house computer programs for analysis of hydraulic systems. 3.1 Pipe systems A simple model of steady flow in pipelines or pipe networks can be created by combining mass conservation principle at pipe nodes (analogously to wellknown Kirchhoff rule) and Darcy-Weisbach formula describing the pressure drop in pipes connected to the nodes. Thus, assuming incompressible fluid flow, the following relations apply: where i and j are the indices of the nodes and pipes, respectively, n i is the number of nodes in the pipe system, n j is the number of pipes, Q is the flow rate through a pipe, A is the area of the pipe cross section,

p is the static pressure difference between the pipe

ends, λ is the Darcy-Weisbach friction factor, L is the

length of the pipe, and D is its inner diameter.

Requiring that Equation 4 is fulfilled for every pipe

node i, and adopting at the same time that Equation

6 holds for every pipe j connected to the node under

consideration, one obtains the system of equations of the form:

where a_{ij} are the coefficients of the matrix of the system of equations and b_i are the coefficients of the source array. This system can be solved iteratively for the nodal pressures p_i . Having known the nodal pressures, the liquid velocity in the pipes is easily obtained from Equation 6, and herewith the flow rates .

3.2 Surge tank

A model of water flow in a surge tank with a side chamber may be obtained from the laws of conservation of mass for the tank, conservation of mass for the junction of the surge tank and the headrace tunnel, and conservation of linear momentum for the headrace tunnel, which are given in respective order:

where A_{fs} is the area of the water free surface (in a simply-shaped tank it is just the area of the tank cross section), z is the vertical displacement of the water free-surface in the tank, A_c and v_c are the cross-section area and average water velocity in the junction, A and v are the cross-section area of the tunnel and the corresponding water velocity, v_{out} is the water velocity on the exit side of the tunnel (towards the penstock and the turbines), g is the gravity acceleration, L and D refer to the headrace tunnel (not to the tank), and

ξ_c is the minor loss coefficient in the junction region relating the local static pressure drop and the dynamic pressure in the junction $\Delta p_c = \xi_c \rho v_c^2 /2$.

When the losses are neglected and the areas are constant, Equations 8–10 can be solved analytically, yielding the water elevation in the form of a periodic sine function. In this work, Equations 8–10 are solved numerically for z and v using implicit Euler scheme for approximation of the time derivatives. In the case of a surge tank with a side chamber, a correction of A_{fs} in Equations 8, 10 is done taking into account its considerable increase when the water fills the cham ber during the free-surface rise, see Figure 1. By this solution, one obtains lower water elevation and longer oscillation periods than in the case without the side chamber. Figure 1. Typical motion patterns of water in a surge tank with a side chamber (a)-(d), shown in chronological order. The water (grey-shaded area) enters the tank from the headrace tunnel which is connected to the tank at its bottom side. Figure 2. Static pressure variation (grey-shaded) along a coiled pipe modeled as a set of straight pipe elements. Note that, since the calculation is based on integral 1D model, the element thickness shown here does not represent the pipe diameter. 4 EXAMPLES 4.1 Flow in a helically coiled pipe Integral model described by Equations 4–6 can be effectively used for calculation of steady liquid or incompressible-gas flows in pipe networks and pipelines consisting of straight or slightly curved elements. Substitution of a strongly curved pipe by a set of discrete, straight sections, even in case of a large number of short such elements, would not deliver acceptable results. See Figure 2 for an example where the geometric model and the static-pressure variation along the pipe are shown, with flow direction from the left to the right.The physics captured by the discretized geometric model does not recognize curvature effects, and the resulting pressure drops are notably lower than in reality. The simplest and effective solution to this issue is the use of modified values of the friction factor based on empirical data, such as proposed by Ito (1959). Doing so, one obtains the expected pressure drop values in relatively short time. However, the accuracy depends on the empirical friction factor relation directly applied, not on resolving physical effects. The helical pipes are used in a variety of technical applications (e.g. in heat exchangers, where heat transfer is the most important phenomenon, and even phase change may occur), requiring thus more sophisticated solution methods applicable to complex physical problems including hydraulics. In that case,

Figure 3. Friction factor in a helical coil pipe: comparison between CFD-calculated and empirical values.

CFD provides a good framework for analysis. Hence, CFD solution is also tested. The pressure drop in the helically coiled pipe depicted in Figure 2, with curva ture D/D coil = 0.25 where the D is the inner diameter of the pipe and D coil is the mean diameter of the coil, is calculated for a range of Re-numbers cover ing laminar, transitional and turbulent flow regime. The corresponding values of Darcy-Weisbach friction factors are then evaluated. Figure 3 shows the result ing values. Laminar CFD solution demonstrates good agreement with both available empirical solutions in laminar flow regime (White 1932, Ito 1959), and also slightly beyond the transition point between laminar and turbulent flow. Interestingly, also k-ω SST tur bulence model predicts friction factors well both in laminar and transitional regime in which the relations of Ito (1959) and Gnielinski (1986) are approached.

Its results for higher Re-numbers are within the band determined by the presented empirical relations, where agreement with the relation given by Prandtl (1949) is the best. Realizable k-ε model however demonstrates large deviations in transitional regime, as expected, but at high Re-numbers is close to the relations of Ito (1959) and Gnielinski (1986).

4.2 Flow in a surge tank of a hydropower plant

Problem of water elevation in a surge tank of hydropower plant Jablanica in Bosnia-Herzegovina, for which limited measured data are available, is con sidered. The subject of analysis is the case of turbine wicket-gate closure triggering water hammer effects, which subsequently cause water level rise in the tank followed by damped oscillations of the water column in the tank.

For a comparison with the available measured data, two different flow rates are taken into account: 84.5 m^3/s with initial position of the water free sur face in the tank at 51 m above the axis of the headrace tunnel (268 m above sea level), and 72 m^3/s with the initial position of the water surface in the tank at 44.4 m above the headrace tunnel. Figure 4. Computational mesh around the junction of the headrace tunnel and the surge tank (above). Pressure distribution calculated for the nominal flow rate of 90 m^3/s from the headrace tunnel to the tank: using k-ω SST model (below, left) and using realizable k-ε model (below, right). Through numerical testing it was found that the surge-tank model performance depends strongly on the input value of the minor-loss coefficient describing the energy loss in the region around the surge tank junction. Detailed analysis of this region was inevitable. Hence, a CFD study of the water flow from the headrace tunnel to the tank and in reverse direction is conducted in order to estimate the pressure loss precisely. Note that at the bottom of the tank a diaphragm for additional damping is mounted. In order to capture its impact onto the flow resistance additional mesh refinement in this critical region is done. The computational mesh used, with local cell refinement in diaphragm region and prism layers along the walls for proper capturing of boundary layer, and pressure distribution are shown in Figure 4. Averaged pressure drop between the indicated monitoring sections is calculated. The corresponding results of the minor loss coefficient from calculations with two models of turbulence are shown in Table 1. Calculation with k-ω SST model delivered the values of the minor loss coefficient about 20 on average for the case of flow into the tank, and it is slightly higher (about 10%) for the reverse direction. The pressure drop results obtained with realizable k-ε model do not differ considerably, but the calculation is more stable, as expected, and slightly faster. Having passed the values of the minor-loss coefficient calculated using CFD on to the computer program with the integral model for surge-tank flow, Equations 8–10, the time-dependent position of the

Table 1. Minor loss coefficient in the junction region.

Minor loss coefficient ξ_c	k-ω SST model	Two-layer, realizable k-ε model
Flow from the tunnel to the tank	20.2	19.5
Flow from the tank to the tunnel	22.2	22.5

Figure 5. Time history of the water level in the surge tank:

for the flow rate of 72 m^3/s with the initial water level 44.4 m (above), and for the flow rate 84.5 m^3/s with the initial water

level 51 m (below).

water free-surface in the surge tank is calculated, which

is shown in Figure 5 by thick solid line. The analyti

cal solution for cylindrical tank, whose cross-section

is modified to take into account the volume of the side

chamber (dashed line), and the results of a detailed

CFD analysis from a previous study (Torlak et al.

2015), thin solid line, are shown for comparison as

well. The analytical solution is obtained from Equa

tions 8-10 for simplified, cylindrical geometry and

ignoring the loss terms (those including λ and ξ_c). It

results in periodic variation of the water level with sta

tionary amplitudes. The amplitudes are considerably

higher than expected, since the losses are not taken

Using Unstructured Moving Meshes with Cells of Arbi

trary Topology. Comput. Meth. Appl. Mech. Engrg. 125:

235–255.

Ferziger, J.H. & Perić, M. 2013. Computational Methods for

Fluid Dynamics. Springer.

Gnielinski, V. 1986. Correlations for the Pressure Drop in

Helically Coiled Tubes. Int. Chem. Eng. 26 (1): 36–44.

Ito, H. 1959. Friction factors for turbulent flow in curved

pipes. J. Basic Engineering 81: 123–134.

Menter, F.R. 1994. Two-equation eddy-viscosity turbulence modeling for engineering applications. AIAA Journal 32 (8): 1598–1605.

Muzaferija, S. & Peric´ M. 1998. Computation of free surface flows using interface-tracking and interface-capturing methods. In O. Mahrenholtz & M. Markiewicz (eds), Nonlinear water wave interaction, Chap. 3. Southampton: Computational Mechanics Publications.

Muzaferija S. & Peric´ M. & Sames P. & Schelin T. 1998. A two-fluid Navier-Stokes solver to simulate water entry. In Proc. Twenty-Second Symposium on Naval Hydrodynamics. Prandtl, L. 1949. Fuehrer der Stroemungslehre, 3rd edition, Braunschweig: Vieweg. Rodi, W. 1991. Experience with Two-Layer Models Combining the k-ε Model with a One-Equation Model Near the Wall. In 29th Aerospace Sciences Meeting, 7–10 January 1991, Reno, NV, AIAA 91-0216. Shih, T.H. & Liou, W.W. & Shabbir, A. & Yang, Z. & Zhu, J. 1994. A New k-ε Eddy Viscosity Model for High Reynolds NumberTurbulent Flows – Model Development and Validation. NASA TM 106721. Torlak, M. & Bubalo, A. & Džaferovic´, E. 2015. Numerical analysis of water elevation in a hydropower plant surge tank. In Proc. 6th IAHR International Meeting of theWorkgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems, 9-11 September 2015. Ljubljana, Slovenia. White, C.M. 1932. Fluid friction and its relation to heat transfer. Trans. Inst. Chem. Eng. 10: 66–86. London. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Seepage characteristics in embankments subject to variable water storages on both sides

M. Calamak

Department of Civil Engineering, TED University, Ankara, Turkey

A. M. Yanmaz

Department of Civil Engineering, Middle East Technical University, Ankara, Turkey

ABSTRACT: Long embankments are constructed on plains to support highways and railways. Depending on

the magnitude of the flood volume, topographical characteristics of the plain and surface coverage of the flood

over the plain, embankments may be subject to high water levels on both sides. With the construction of series of

culverts through the embankment, accumulated water on one side may transmit to the other side. In this study,

effects of unsteadiness of water levels in both sides of a sample embankment are investigated with reference to

seepage pattern throughout the embankment.Analyses are conducted using the software SEEP/W ® . The findings

of the analyses provided means to assess the hydraulic performance of the embankment in view of suitability of

the soil characteristics and layout of the layers forming the embankment.

1 INTRODUCTION

Flood inundation areas may cover large zones over flat

plains during the occurrence of high frequency floods.

Long embankments constructed to support highways

and railways on such terrains may be subject to high

water levels on both sides. Degree of water rising on

the plain is dictated by the size and number of outlet

facilities, such as culverts through the embankment as

well as the surrounding topographic features. The flow

from the one side of the embankment to the other is allowed by a series of culverts and bridges.

A typical highway or railway embankment resting on a flat plain can be subject to various external loadings during its lifetime. These loadings include dynamic loads originating from the vehicles travelling above the embankment, earthquakes and floods having high return periods. The common modes of failure in an embankment are attributed to overtopping of the crest by a flood wave, occurrence of excessive seepage through the embankment during a flood, failure of the slopes, and the reduction of the crest level due to excessive settlement (Morris et al. 2007).

The excessive raising of inundated water in the upstream side of the railway embankment is normally resulted from inadequate capacity of bridge openings and culverts in transmitting the flow. At the same time, the water transmitted to the downstream of the embankment may not flow easily due to relatively small slope and restrictions induced by topographic features and land use characteristics. These conditions would then yield time-dependent raising and lowering of water levels on both sides of the embankment during the passage of a severe flood. The existence of water in both sides will cause some quantity of water flow through or beneath the embankment since the body and the foundation of the structure are made of porous materials. Excessive seepage can cause movement of the finer particles, and progressively making the embankment more permeable and resulting in increased seepage rates. Eventually, this may cause piping situation which may lead to the failure of the structure. The seepage can also increase the probability of failure due to the slope instabilities. The increased pore water pressures in the embankment reduce the factor of safety against sliding of the slopes. Therefore, both the seepage rate and the pore water pressures should be estimated under various flooding conditions, which are compatible to the local hydro-meteorological and topographic conditions. Several researchers focused on railway embankments related with their design using empirical approaches, finite element models and neural networks (Woldringh and New 1999; Egeli and Usun 2012; Tayfur and Egeli 2013), deformation analysis (Mu et al. 2015), temperature change and settlement (Wei et al. 2006), reinforcement with piles (Zhuang and Cui 2015) and geogrids (Montanelli and Piergiorgio 2003). A seepage-related study was conducted by Chen et al. (2014) which dealt with determination of the seepage coefficient and the hydraulic gradient of the embankment made of volcanic cinder gravels with site tests. An application study is needed to understand the seepage pattern and possible seepage related problems in railway embankments. This study investigates the seepage characteristics of embankments under flooding conditions. A typical embankment is defined in a hypothetical application

and modeled for seepage under transient boundary

conditions at its both sides. The change of the flux

through and beneath the structure, the maximum

hydraulic gradients and the pore water pressures with

respect to time are determined in the embankment.

Conclusions are drawn about the seepage behav

ior and some geotechnical issues for embankments

constructed on a flat plain.

2 SEEPAGE MODELING

The seepage through embankments subject to floods

can be modeled using Darcy’s law defining the gov erning differential equation of the flow (Richards 1931):

where H is the total head, K_x and K_y are the hydraulic conductivities in x and y directions, respectively, Q' is the external boundary flux, θ is the volumetric water content, and t is the time.

The above equation can be solved using appropriate numerical methods and initial and boundary conditions to determine the heads, pore water pressures, and fluxes through and beneath the embankment. One of the most robust techniques for this is finite element method (FEM). In the present study, the seepage problem is solved using SEEP/W ® adopting FEM.The solution algorithm of the software is provided in detail in Geo-Slope Int Ltd (2013) and Papagianakis and Fredlund (1984).

The layers and the foundation of the embankment are assumed to be homogeneous. An unsaturated and saturated soil model is adopted for the cohesive soil layers of the embankment. The properties of the soil water characteristic curves are estimated using van Genuchten method (van Genuchten 1980). The unsaturated hydraulic conductivities are determined using this method with the following relationships:

Table 1. Material properties of the embankment layers. Ks α θ s r θ r

Layer	(m/s)	(cm $^{-1}$)	n	(m ^{3}/m ^{3})	(m ^{3}/m ^{3})
Ballast	1.00×10^{-1}	-	-	0.50	-
Subballast	1.00×10^{-3}	-	-	0.40	-
Subgrade	1.00×10^{-4}	-	-	0.20	-
Variable fill	2.20×10^{-6}	0.027	1.23	0.38	0.100
Natural ground	1.28×10^{-6}	0.008	1.09	0.38	0.068

where, K_r, K_s and K are the relative, saturated and computed hydraulic conductivities, respectively, and the variables α (cm $^{-1}$), n and m are fitting parameters of the curve and m is related with n by $m = 1 - 1/n$.

3 APPLICATION PROBLEM AND ANALYSIS

A hypothetical application is presented herein. It consists of a typical railway embankment. The route of the railway intersects a river and its tributaries at many locations where the flow is provided with culverts and bridges. During a severe flood condition the water cannot be transmitted to the downstream effectively and it starts to accumulate in front of the embankment.At the same time, due to the slightly small slope of the basin it also starts to accumulate at the downstream side. The time-dependent nature of the flood then causes an unsteady flow through and beneath the embankment. The typical cross-section of the embankment is given in Figure 1. The total height of the embankment measured from the natural ground is 7.2 m. The layers of the fill from the top to the ground are ballast, subbalast, subgrade and variable fill. The side slopes of the ballast are 1V:2.0H, whereas those of the variable fill are 1V:3.0H. The ballast, subballast and subgrade layers are made of graded non-cohesive aggregates having certain maximum and minimum particle sizes and resistances. The variable fill is composed of well graded compacted sandy and gravelly clay which is free from organic materials. The coefficient of gradation (curvature) for the variable fill is approximately unity.The natural ground soil type is clay.The geotechnical properties of the layers are given in Table 1. In this table, θ_s and θ_r stand for saturated and residual water contents, respectively. The variation of the maximum possible flow depths at the upstream (U/S) and downstream (D/S) of the embankment are assigned as shown in Figure 2. This information was extracted from a previous case study conducted by the authors. The maximum flow depths measured from the natural ground at the upstream and downstream are 6.8 m and 5.0 m, respectively. Therefore, the embankment is not overtopped during the flood. At the initial stage, prior to the flood, the embankment is assumed to be completely dry.Therefore initial flow depths at the upstream and downstream are considered to be zero and the initial water contents of the soil layers are taken at their residual values.

Figure 1. The cross-section of the railway embankment modelled in the study (Not to scale).

Figure 2. Upstream and downstream boundary conditions

of the embankment.

As can be seen from Figure 2, the effects of the flood take place for less than 2 days and the sim ulation duration of the transient seepage analysis is chosen as 20 days. The modeling of the transient seep age yielded the seepage flux, the pore water pressures and the hydraulic gradients through and beneath the embankment.

4 RESULTS AND DISCUSSIONS

The distribution of the pore water pressures through and beneath the embankment at three time steps of the simulation, which correspond to rising and falling stages of the flood (t = 8.8 hrs and t = 18.2 hrs) and after the flood (t = 182.2 hrs), are presented in Fig ure 3. Also, the variations of seepage fluxes passing through and beneath the embankment at the center line, and at the upstream and downstream faces are provided in Figure 4. Figure 3. The phreatic surface and the pore water distribution at (a) t = 8.8 hrs; (b) t = 18.1 hrs; (c) t = 182.2 hrs (Pore water pressures are in kPa).

Figure 4. The variation of the seepage fluxes at different locations of the embankment.

As can be seen from Figure 3, with the increase in the water level at the upstream and downstream sides, the wetting lines of the seepage move towards the cen ter of the embankment from both sides. When the flow depth at the upstream reaches its maximum value, the greatest pore water pressures are observed at this side (see Figure 3(a)). Similar conditions occur at the down stream side when the flow depth here is maximum (see Figure 3(b)). After a considerably long time, the satu rated parts of the embankment are completely drained and smaller pore water pressures are observed at the embankment (see Figure 3(c)). Since the flood water is drained from both sides in a relatively short time, the central part of the embankment stays dry during the flood.

According to Figure 4, seepage rate at all sections, except beneath the embankment, increases during the rising stage of the flood. They reach their maximum values when the water level at the upstream is at its maximum value (t = 8.8 hrs). The maximum flux is observed through the upstream face of the embank ment with a value of 1 m^3/hr. Then with the decrease in the inflow of the flood, the seepage rates start to decrease across the embankment.The flow is observed to completely stop after 40 hrs. It is seen that the fluxes occurring beneath are insignificantly small for all times.

To assess the stability the embankment against piping, the maximum hydraulic gradients across the embankment are determined and compared with the critical hydraulic gradients of the soil layers. The critical hydraulic gradient, which is the ratio of the submerged unit weight of the soil to the unit weight of water (Terzaghi et al. 1996), is defined as the gradient at which the particle motion is initiated (Skempton and Brogan 1994). When the maximum hydraulic gradient exceeds the critical one, the soil particles start to be washed out of the embankment.

The change of the maximum gradient across the embankment with respect to time is presented in Fig ure 5. In this figure, the points defining the curve are numbered. These points refer to various locations in the embankment and they are defined in Table 2. Also,

the critical hydraulic gradients of two soil layers which Figure 5. The change of the maximum gradient in the embankment with respect to time. Table 2. The locations of the points where the maximum gradients are observed. Maximum gradient point Location 1, 2, 3, 4, 5, 6 Variable fill 7, 8, 9, 10, 11, 12, 13, 14, 15, 16, 17, Natural ground 18, 19, 20 accommodate the maximum gradient points, are indicated on Figure 5 as threshold lines. These layers are the variable fill and the natural ground which have the critical hydraulic gradient values, 1.48 and 1.14, respectively. The results showed that the maximum hydraulic gradient locations are varying within only two layers. It becomes maximum when t = 5.3 hrs with a value of 2.05 inside the variable fill layer.The critical hydraulic gradient is exceeded by 39% at this instant. For a total duration of 10.6 hrs, the maximum hydraulic gradient is observed to exceed the critical hydraulic gradient (see Figure 5) at some certain locations. These are at the upstream side of the embankment and rest inside the variable fill layer. This happened during the rising stage of the flood at which the flow depth at the upstream increased and the wetting front of the seepage propagated towards the center of the embankment. For this time period there is a high possibility of movement of soil particles towards the center of the embankment. However, this will not cause piping since the water at the upstream face has drained and the propagation of the seepage wetting line stopped in the following hours. After the water is totally drained from both sides of the embankment (t = 35 hrs), the maximum hydraulic gradients are observed only in natural ground and all gradients are smaller than the critical gradient of the soil of this part. In view of the embankment safety, slope stability analyses are also conducted in the scope of the study. The pore water pressures through the embankment body vary with time during a flood. This results

Figure 6. The change of factor of safety against sliding for both upstream and downstream slopes.

in variation of the factor of safety against sliding for both the sides of the embankment. Especially during the drawdown of the flood, it is possible to observe factor of safety values less than unity (Calamak et al. 2015). To this end, using limit equilibrium analysis, the change of factor of safety values against sliding are assessed for upstream and downstream slopes of the application problem.Analyses are conducted using the software SLOPE/W ® . The results of the analyses are presented in Figure 6 for upstream and downstream slopes of the railway embankment. The results showed that the minimum factor of safety values are approxi mately 1.70 and 1.65 for the upstream and downstream slopes, respectively.The minimum factor of safety val ues are shown to be greater than the minimum required value which is 1.50.

5 CONCLUSIONS

This study investigated the seepage behavior of embankments constructed on flat plains to support highway and railways during flooding conditions.

Such embankments may be subject to water level vari ations on both sides, which normally differentiate them from levees and embankment dams.Also their material composition differ them from other earthen structures.

In the application of the study, a hypothetical railway embankment to be constructed on a flat plain is con sidered and its seepage characteristics are investigated under a synthetic flood. The hydraulic boundary con ditions were defined arbitrarily and the analyses were conducted using a FEM software.

The results of the analyses showed that the seepage wetting line propagated to the center of the embank ment during the rising stage of the flood from both sides. During this period, the pore water pressures at the upstream and downstream sides and at the founda tion of the embankment increased. Also, the seepage flux passing through both faces and the centerline of the embankment increased. Then, during the falling stage, both the pore water pressures and the seep

Canadian Geotechnical Journal, NRC Research Press, 21(3), 419–430.

Richards, L. A. (1931). "Capillary conduction of liquids through porous mediums." Journal of Applied Physics, 1(5), 318–333.

Skempton, A. W., and Brogan, J. M. (1994). "Experiments on piping in sandy gravels." Gẽotechnique, 44(3), 449–460.

Tayfur, G., and Egeli, İ. (2013). "Railway embankment design based on neural networks." GRAĐEVINAR, 65(4), 319–330.

Terzaghi, K., Peck, R. B., and Mesri, G. (1996). Soil mechanics in engineering practice. John Wiley & Sons Inc. Wei, J., Xu, Z. Y., Yang, C. Y., and Xu, M. X. (2006). "Study onTemperature and Settlement of anAir-Cooled Embankment at the Qinghai-Tibet Railway." Advances in Earth Structures: Research to Practice, Shanghai, 286–293. Woldringh, R. F., and New, B. M. (1999). "Embankment design for high speed trains on soft soils." Geotechnical Engineering for Transportation Infrastructure, 1–10. Zhuang, Y., and Cui, X. (2015). "Case Studies of Reinforced Piled High-Speed Railway Embankment over Soft Soils." International Journal of Geomechanics, American Society of Civil Engineers, 6015005. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Characterization of nappe vibration on a weir

M. Lodomez, M. Pirotton, B. Dewals, P. Archambeau & S.

Erpicum

University of Liège, Argenco Department, Liège, Belgium

ABSTRACT: The nappe vibration phenomenon may affect numerous prototype structures, such as gates and

spillways, when they operate under low head conditions. This phenomenon, which is far not well understood

and poorly controlled, produces frequency oscillations in the thin flow nappe cascading downstream of the

weir. These oscillations result in a disturbing noise production that increases the environmental and societal

impacts of the hydraulic structure. Given limited information available regarding the physical processes of this

phenomenon, a detailed investigation has been undertaken to characterize the flow for free-overfall structures

where nappe vibration may be of concern. The research conducted on a prototype-scale linear weir model (weir

length of 3.5 m and a fall height of 3 m) at the Engineering Hydraulics laboratory of the University of Liège

enable to describe the flow and nappe vibration by using image and sound analysis. The paper presents the first

quantitative results of the study.

1 INTRODUCTION

As a key component of dams safety, weirs are com

monly used as spillways to release flows from a

reservoir.The free falling jet on the downstream side of

the weir, called nappe, can display a variety of behav

iors as developed for labyrinth weir in Crookston &

Tullis (2012b). Under relatively low-head discharges,

the behavior of nappe oscillation, otherwise known as nappe vibration, can occur. This phenomenon classi fied by Naudascher & Rockwell (1994) as an insta bility induced excitation, has been also observed on hydraulic structures with a free overfall, such as crest gates and fountains. Early identified as a undesir able and potentially dangerous, the most recognizable characteristics of this phenomenon are the horizon tal banding on the nappe and the extreme acoustic energy resulting in low frequency noise that can be hear in close proximity of the structure (Casperson 1993; Crookston et al. 2014).

In the case of flow over a gate, the occurrence of the phenomenon has been attributed in part to the inter action between the flow and the enclosed air pocket between the gate and the nappe (Naudascher & Rock well 1994). In fact, the implementation of splitters to the gate crest in order to divide the water falling sheet and aerate the air pocket has proven to be an effective mitigation technique and represent today a good prac tice in gates design (Naudascher & Rockwell 1994; Sumi & Nakajima 1990; USBR 1964). However, for free surface weirs, the nappe vibration is observed even with an aerated air pocket behind the nappe (Bin nie 1972; Crookston et al. 2014; Crookston & Tullis 2012a; Falvey 1980; Schwartz 1966). Nevertheless, for these structures, effective mitigations techniques exist. Indeed, Metropolitan water, sewerage and drainage board (1980) reports the effectiveness of roughness modification. Lodomez et al. (2016) asses and optimizes practical countermeasures (crest modifications) to nappe vibrations. From a scientific point of view, the physical processes of this complex flow behavior are still unclear and a lack of consensus exists on the causes and source of the oscillations. The most suggested theory is the Kelvin-Helmholtz mechanism (Helmholtz 1868) that attributes nappe instabilities to shear forces that occur at the interface between the falling water and air. This theory therefore requires an appreciable velocity differential between the two fluids, which is likely not reached at the crest of a weir. Casperson (1993) describe the resonance with the air pocket entrapped behind the nappe as a cause of amplification of nappes oscillation while Chanson (1996) suggest a pressure discontinuity at the weir crest as the cause of the phenomenon with an origin of the vibrations at the crest. Recently, new investigations have been undertaken at the Utah Water Research Laboratory (Utah State University) (Anderson 2014; Crookston et al. 2014). They indicate that the initiation of the instability most likely occurs at the weir crest. Indeed, the horizontals stripes resulting in the vibrations are visible directly after the flow separation from the weir crest (even for vented nappe models). Moreover, it has been proven that roughness modification of the weir affects the vibration. These recent observations with some conflicting results from previous analysis confirm the need for systematic data measurement on large-scale models to come up with generic scientific conclusions as well as a deeper systematic analysis of scale effects.

In this context, the current experimental study was undertaken to develop generic scientific conclu sions regarding the physical processes by means of an experimental study on a large-scale model. This paper provides an overview of the preliminary results regarding the characterization of the nappe vibrations.

2 EXPERIMENTAL SETUP

A detailed investigation of nappe vibration is being conducted at the Engineering Hydraulics laboratory of the University de Liège.The study utilizes a prototype scale linear weir.As illustrated in Figure 1, the physical model is an elevated box divided in two identical weirs with a 3.46-m long crest and a 3.04-m high chute.

The air cavity behind the nappe can be confined or vented to the atmosphere. Indeed the weir located on the left bank is confined between two lateral walls and a back wall, one lateral confinement wall being transparent (Plexiglas) and others black multiplex panels. The other model is vented except on the shared lateral side. The models are used independently by obstructing with multiplex panels the unwanted crest.

The weir crest geometry is a quarter round crest (15-cm radius and 15-cm long flat element) which is modelled with typical prototype dimensions of reinforce concrete weir.

Flow is supplied to the model by means of two pipes, connected to two regulated pumps. As illustrated in Figure 2, flow gets into the model through perforated pipes, which are parallel to the crest and located on the bottom of the reservoir. The discharge is measured with an electromagnetic flow meter installed on the supply piping. The maximum unit discharge is 7.22 10^{-2} m^2/s (250 l/s in the model). In order to provide flow velocities as uniform as possible upstream of the crest, a baffle wall of synthetic membranes is inserted into the reservoir (Figure 2).

To identify and quantitatively characterize the nappe oscillation, a microphone and high-speed video cameras have been setup. The microphone placed in front of the falling nappe and its operating software

Figure 1. Prototype-scale model. supply, after a Fourier transform of audio signal, a characterization of the phenomenon in terms of sound frequency and amplitude. The high-speed video camera is used to capture images of the falling water. The acquisition frequency of camera is either 240 Hz or 300 Hz, depending of the camera. Image analysis used to quantify the nappe oscillation frequency is based on the detection of horizontal bands in the falling water. 3 RESULTS AND DISCUSSION 3.1 Sound analysis The first quantitative results obtained based on acoustic measurements are audio spectrums which display the dominant frequencies and their sound level (Lodomez et al. 2016). For a range of unit discharges, audio spectrums of sound recording are illustrated in Figure 3–6 for the confined and the vented model. In Figure 2. Upstream device: reservoir, water supply pipes and baffle wall. Figure 3. Audio spectrum of sound recording for 0.015 m 2 /s. Figure 4. Audio spectrum of sound recording for 0.03 m 2 /s.

addition, the dominant frequency and the associated sound level are reported as a function of the discharge in Figure 7.

As a first observation, it is observed that there is a dominant peak frequency whatever the model for unit discharge between 0.015 m 2 /s and 0.05 m 2 /s. The comparison between the confined and vented model show that the sound level of the dominant peaks are higher for the confined model. This is in agreement

with the assumption of Casperson (1993) regarding

the amplification effect of the confinement.

Then, for the confined model, it can be noticed

that the dominant frequency varies between 30.75 Hz

Figure 5. Audio spectrum of sound recording for

0.045 m^2/s.

Figure 6. Audio spectrum of sound recording for 0.06 m^2/s.

Figure 7. Summary of sound measurements. Continuous

lines represent nappe vibration area with regard to the

combination of high sound level and constant dominant

frequency. and 35.5 Hz, for unit discharges between 0.01 m^2/s and 0.07 m^2/s. For the vented model, the dominant frequency seems more variable. In fact, a constant dominant frequency of 35.5 Hz between 0.025 m^2/s and 0.04 m^2/s is observed while higher frequency of the order of 45–50 Hz are measured for unit discharge lower than 0.025 m^2/s and higher than 0.04 m^2/s.These results show for each model, a range of constant dominant frequency associated with a significant sound level (higher than 80 dB). As mentioned in Lodomez et al. (2016), the combination of high sound level and constant dominant frequency documents a unit discharge range of the nappe vibration phenomenon. Therefore, the nappe vibration phenomenon is shown to be dominant, and disturbing, between 0.015 m^2/s to 0.05 m^2/s, and between 0.025 m^2/s to 0.04 m^2/s, respectively for confined model and vented model (continuous lines in Figure 7). 3.2 Image analysis As already mentioned, horizontal bands are the visible characteristic of nappe oscillation phenomenon. Therefore, in conjunction with sound measurement, flow visualizations for various flow rates and both model configurations, have been conducted. Figure 8 shows the images obtained for the confined model. Figure 8. Nappe vibration visualization for (a) 0.015 m^2/s, (b) 0.03 m^2/s, (c) 0.045 m^2/s and (d) 0.06 m^2/s for the confined model.

Horizontal bands are clearly visible for unit discharge lower than 0.06 m 2 /s, which is in agreement with sound measurements. In fact, the flow visualization shows nappe vibrations for the same flow ranges as those derived from the sound analysis. Tracking of parti cles in the nappe flow attests that particle of falling water move in a continuous arcing path, as opposed to a back and forth oscillation of particles. This obser vation, confirmed by Anderson (2014) is the basis of the image analysis method presented in this paper and also suggests that the instability is created at the down stream edge of the crest and causes a periodic variation of nappe trajectory.

Assuming that the horizontal bands are due to the lighting on the undulating surface, the frequency of the horizontal bands has been determined by the extrac tion of information carried by a fixed set of pixels on a succession of images. The method is concep tually sketched in Figure 9. The undulating nappe surface, represented at three successive time steps,

Figure 9. Conceptual representation of image analysis.

Figure 10. Sequence of 20 images (camera acquisition rate of 300 Hz). shows horizontal bands (lit or unlit bands) that move according to the flow direction.Then, for a fixed image frame, a chosen line of pixel carries information which varies depending on whether the pixels are on the saturated zone of the oscillation or not. This information is translated on each pixel of the line as a numerical value between 0 and 255 (images encoded in 8 bits). In order to illustrate the first step of the image analysis, Figure 10 shows a set of 20 spatially fixed frames and

several chosen lines of pixels (in dashed lines). It is understood that the camera acquisition rate has to exceed the nappe vibration frequency. In Figure 10, the course of the oscillations can be tracked and shows that two oscillations passed through the frame during the acquisition time of 20 images. The extracted information from a chosen line (300 pixels per width of the frame) is represented in Figure 11 (a) for each image, i.e. in time. Based on Figure 11 (a), the second step of the method is the detection of the number of waves passing through the line. It has been implemented using the mean value of the line of pixels (Figure 11 (b)). Finally, the frequency of the visible oscillation, F_v is calculated as: where N_w =number of detected wave; F_f =acquisition frequency of the camera; and Nf =number of frame. The accuracy of the image frequency calculation depends of the ratio between F_f and N_f . . For an acquisition frequency of 300 Hz (resp. 240 Hz), a minimum of 1200 frames (resp. 960 frames) have to be used to ensure an accuracy of 0.25 Hz, similar to that of the sound measurements.

Figure 11. (a) Extracted lines of pixels from the sequence

of 20 images. (b) Evolution of the mean value of the extracted

line of pixels.

For confined and vented model, the image analysis has been applied to unit discharges for which horizontal stripes can be observed. The number of chosen pixels lines is at least 5 for each set of images. The number of sets of analyzed images is between 3 and 5 per unit discharge. The results are shown in Figure 12. For the vented configuration, it is observed that a frequency (around 35 Hz) is obtained in the range between 0.025 m^2/s and 0.045 m^2/s. The frequencies obtained from image analysis do not differ by more than 2.75 Hz. For a discharge of 0.02 m^2/s and

0.05 m 2 /s, frequencies are in the order respectively of 50 Hz and 48 Hz and are more variables. For the con fined configuration, all results are between 34.5 and 31.25 Hz. All results demonstrate that the frequency is globally constant in space, i.e. along the nappe. Indeed, the frequency variation calculated between different lines, spaced along the falling nappe, is of the same order as the frequency variation calculated between different set of images. Finally, as already mentioned in the sound analysis, a range of frequency between 32 Hz to 35 Hz seems to characterize the nappe vibration phe nomenon for both configurations and a common flow range.

3.3 Frequency comparison

The comparison of the frequency of nappe vibration phenomenon determined by image and sound analysis is illustrated in Figure 13, in the common flow range of vibration. This comparison clearly shows that the Figure 12. Results of image analysis. Figure 13. Comparison of frequency obtained using image and sound analysis. sound frequency is identical to the frequency of horizontal bands, in the range between 31 Hz to 37 Hz.As a consequence of this finding, the noise generated by the phenomenon cannot be explained by the impact of the nappe on the ground. Indeed assuming that the cause of noise is due to the impact, the sound frequency should be twice the visual frequency since the impact varies between two opposite particle of the cyclical nappe waves. This result is in agreement with the observa tion ofAnderson (2014) according to which the impact surface has little effect on nappe vibrations.

The mean value of vibration frequency for confined and vented model, represented by the dashed lines, suggests that the frequency of vibration is slightly different depending of the confinement.

4 CONCLUSIONS

The research conducted on a prototype-scale linear weir model enable to describe the nappe vibration by using image and sound analysis, especially in terms of frequency vibrations. The main result is the obvious link between the frequency of the sound and the frequency of horizontal stripes in the thin flow nappes. Indeed, these frequencies are identical in the range of 31 Hz to 37 Hz, for a prototype-scale linear weir with a 3.46-m long crest and a 3.04-m high chute. Moreover, the current experimental study indicates that the frequency vibration range is constant, for a fixed crest shape and whatever the confinement configuration. Similarly the flow range for which vibrations appear is identical in both configurations.

The study is still ongoing with additional tests to determine potential parameters that characterize the initiation of nappe vibration such as upstream hydrodynamic conditions, the final aim being to determine the cause of vibration.

ACKNOWLEDGMENTS

The authors are thankful to the C.E.D.I.A., a Techno

logical Support Cell of the University of Liège, for

providing the necessary facilities for sound recording

and processing.

Anderson, A.A. 2014. Master thesis: causes and counter

mesures for nappe oscillation. Utah State University.

Binnie, A. 1972. The stability of a falling sheet of water. In

Proceedings of the Royal Society of London A: Mathe

matical, Physical and Engineering Sciences. The Royal

Society, pp. 149–163. Casperson, L.W. 1993. Fluttering fountains. Journal of Sound and Vibration, 162(2), pp.251–262. Chanson, H. 1996. Some hydraulic aspects during overflow above inflatable flexible membrane dam, Department of Civil Engineering, The University of Queensland, Australia, Report CH47/96. Crookston, B. & Tullis, B. 2012a. Hydraulic design and analysis of labyrinth weirs. I: Discharge relationships. Journal of Irrigation and Drainage Engineering, 139(5), pp.363–370. Crookston, B. & Tullis, B. 2012b. Hydraulic design and analysis of labyrinth weirs. II: Nappe aeration, instability, and vibration. Journal of Irrigation and Drainage Engineering. Crookston, B.M., Anderson, A., Shearin-Feimster, L. & Tullis, B. P. 2014. Mitigation investigation of flow-induced vibrations at a rehabilitated spillway. In ISHS 2014 – Hydraulic Structures and Society – Engineering Challenges and Extremes: Proceedings of the 5th IAHR International Symposium on Hydraulic Structures. Falvey, H. 1980. Bureau of reclamation experience with flowinduced vibrations. Practical Experiences with Flowinduced Vibrations (ed. E. Naudascher & D. Rockwell). Springer. Helmholtz, H. 1868. Ueber die Thatsachen, die der Geometrie zum Grunde liegen. Nachrichten von der Königl. Gesellschaft der Wissenschaften und der Georg-AugustsUniversität zu Göttingen, 1868, pp.193–221. Lodomez, M. Crookston, B. M., Tullis, B. P., Pirotton, M. & Erpicum, S. 2016. NappeVibration MitigationTechniques for Free-overfall Structures. In ISHS 2016 – Hydraulic Structures and Society – Engineering Challenges and Extremes: Proceedings of the 6th IAHR

International Symposium on Hydraulic Structures. Metropolitan Water, Sewerage and Drainage Board 1980. Investigation into spillway discharge noise at Avon Dam. Ancold bulletin, (57), 31–36. Naudascher, E. & Rockwell, D. 1994. Flow-induced vibrations: an engineering guide, Balkema. Schwartz, H.I. 1966. Edgetones and nappe oscillation. Journal of the Acoustical Society of America, 39(3), p.579. Sumi, T. & Nakajima, Y. 1990. Three-dimensional characteristic of nappe oscillation and the estimation of sound pressure levels. Journal of Hydroscience and Hydraulic Engineering, 8(1), pp.89–99. United States Bureau of Reclamation 1964. Experience of the Bureau of Reclamation with flow-induced vibrations, (538), 11–13. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Shock wave patterns in supercritical junction manholes with inlet

bottom offsets

G. Crispino & C. Gisonni

Department of Civil Engineering, Design, Building and Environment, Second University of Naples, Aversa, Italy

M. Pfister

Hydraulic Constructions Laboratory, École Polytechnique Fédérale de Lausanne, Lausanne, Switzerland

Civil Engineering Department, Haute Ecole d'Ingénierie et d'Architecture, Fribourg, Switzerland.

ABSTRACT: Urban floods are frequently provoked by the failure of drainage systems. Sometimes, the mal

functioning of drainage systems arises from uncertain or complicated hydraulic conditions developing into sewer

manholes. Junction manholes represent basic elements for sewers as they enable two or more upstream branches

to merge into a single downstream branch. Initial researches mainly investigated the hydraulic behaviour of

subcritical junctions with constant branch diameters and an even invert. However, the junction of supercritical

flows is the most complex process, mainly due to the likely occurrence of shock waves, breakdowns or choking flows. Furthermore, the sewers laying is commonly characterized by the branch tops alignment. Bottom offsets thus control the manhole inlets when upstream branch heights differ from the downstream one. In this regard, a physical model investigation was carried out at the Laboratory of Hydraulic Constructions (LCH) of the Ecole Polytechnique Fédérale de Lausanne (EPFL). The model reproduced a 45 ∘ junction manhole with aligned up and downstream branch tops and variable upstream branch diameters. The experimental tests focused on the scenario with supercritical flows entering the junction manhole. First outcomes in terms of flow behaviour and shock waves formation are described.

1 INTRODUCTION

Junction manholes are special sewer manholes where two or more upstream branches join into a single down stream branch. Given the pronged configuration of drainage systems, junction manholes are frequently met in sewers. The upstream branches can present different cross-section profiles, dimensions, bottom slopes and outflow elevations.Therefore, the approach flow boundary conditions can be disparate. Shape and dimensions of junction manholes can be still variable, depending on space requirements and drainage net lay out. This multiplicity makes the junction design rather complicate. Thus far, a standard design procedure remains undefined.

As for any sewer manholes, the distinction between subcritical and supercritical approach flows is fundamental. The majority of preceding studies focused on subcritical junctions. Under this regime, the junction design should be aimed to limit head losses and water levels in the manhole. Subcritical merging flows are commonly treated through one-dimensional approach. The junction behaviour can be modelled as a pressurized flow structure if Froude numbers are less than 0.70 (Gisonni & Hager 2012).This hypothesis helped many researchers to compute the local head losses (Idel'cik 1986, Oka & Ito 2005, Hager 2010). If approach flows are supercritical, then the junction flow structure is more complex. In fact, each perturbation of supercritical flows may generate poor combining flow conditions as: – Shock waves in the junction, with choking and air transport interruption of the manhole outlet – Breakdown of supercritical flows resulting in the abrupt transition from free-surface to pressurized flow regime in one or both upstream branches – Upsurging of the manhole, where water level can raise until it blows-off the manhole cover and generates the manhole geysering throughout the worst-case scenario. The design of supercritical junctions targets the safeguard from the occurrence of the above-mentioned failures and the optimization of the manhole discharge capacity. The theoretical approach to model the combination of supercritical flows is particularly challenging. Large flow velocities, two-phase airwater flows and non-negligible head losses require the utilization of the momentum equation. Nevertheless, the definition of the lateral momentum exchange and the wall pressure force components is not trivial for combining supercritical flows (Del Giudice & Hager 2001). For this reason, supercritical junction manholes have been commonly investigated through semi-empirical approaches and by resorting to physical model investigations.

Schwalt & Hager (1995) were the first to study supercritical junctions, by paying attention to shock wave features and choking occurrence for rectangular branches. Del Giudice & Hager (2001) and Gisonni & Hager (2002b) examined shock waves patterns and discharge capacity for circular branches in 45 ∘ and 90 ∘ junction manholes, respectively. Zhao et al. (2004) analyzed the condition of surcharged junction man holes approached by supercritical flows with reference to a case study. Saldarriaga et al. (2012) identified the main shock wave types and the conditions under which waves may occur. In particular, they performed physi cal model tests on a junction manhole with small drops at manhole inlets. Pfister & Gisonni (2014) inves tigated the head losses for supercritical combining flows at 45 ∘ and 90 ∘ junction manholes with circular branches.

All the above-mentioned researches considered junction manholes with equal branch diameters and flat inverts. The upand downstream branch tops were thus at the same elevation. However, if the combining diameters differ, as frequently resulted in urban sewers, then the branch top elevations are diverse. Oppositely, typical sewer concept involves the alignment of the branch tops to avoid backwater effects and to ensure same ground cover on pipe ceilings. In the present study, for the first time the behaviour of a junction manhole with aligned branch tops and different branch diameters was investigated. An extensive experimen tal program was performed on a 45 ∘ junction model under supercritical regime. The branch tops alignment implied the existence of bottom offsets at manhole inlets, given that one or both upstream branch diam eters were fixed to be different from the downstream one. The influence of bottom offsets on the junction flow behaviour required to be analyzed. Prevailing shock wave patterns and poor flow condition occur rences were observed. They are described in detail in the following sections.

2 EXPERIMENTS

2.1 Physical model

Physical model tests were performed at the Labo ratory of Hydraulic Constructions (LCH) of Ecole Polytechnique Fédérale de Lausanne (EPFL). Start ing from the physical model used by Pfister & Gisonni (2014), a new facility was set-up with the aim to align upand downstream branch tops, even for upstream branch diameters different from the downstream one.

The physical model (Fig. 1) consisted of a 45 ∘ junc tion manhole sized similarly to plausible real full-size manhole chambers installed in drainage systems. A straight (subscript o) and a lateral (subscript L) branch with variable diameter D joined in the junction cham

ber, from which a downstream branch (subscript u) Figure 1. Photograph of the junction manhole model under the set-up $D_o = 0.123$ m and $D_L = 0.240$ m (above) and sketch of 45 ∘ junction manhole geometry (below). Figure 2. Junction manholes with: (a) sharp-crested geometry for $D_L = 0.123$ m and (b) and (c) rounded wall geometry for $D_L = 0.190$ m and $D_L = 0.240$ m, respectively. with fixed diameter $D_u = 0.240$ m came out. The junction cross-section was “U”-shaped with bench height of $1.5 \cdot D_u$. The model was supplied by two separate pumps, one for each upstream branch. This arrangement allowed varying the approach discharges Q_o and Q_L, independently. After flowing through the model, water was freely collected in a sump and returned into the laboratory alimentation system. As suggested by Gisonni & Hager (2002a), the downstream part of the junction manhole had a $2 \cdot D_u$ long straight extension (Fig. 1). This addition is aimed to increase the manhole discharge capacity due to a resulted reduction in terms of wave height along the U-shaped cross section. Furthermore, the axial curvature radius was set to $R_a = 3 \cdot D_L$. Therefore, the variation of the lateral branch diameter D_L required the change of the entire junction chamber. The various manholes are shown in Figure 2. As visible, the smallest chamber, corresponding to the minimum lateral diameter, was characterized by a sharp-crested wall geometry at the deflection point between the inner wall of the lateral branch and the manhole bench. Conversely, the remaining two manholes presented rounded wall connections.

Figure 3. Downstream view of the manhole inlets for $D_o = 0.123$ m and $D_L = 0.240$ m.

The branches and the junction were nearly hori zontal so that the bottom slope was supposed to be balanced by friction losses. The limited length of the entire model supports this hypothesis. One of the most important components of the physical model was the jet box. The jet box (Schwalt & Hager 1992) is a sandwich-type structure inside of which a plate char acterized by a given open circular cross-section filling ratio can be inserted. A flow straightener was placed downstream of the predetermined filling ratio plate to improve the approach flow conditions. In the present study, the jet box was placed upstream of both straight and lateral branches. It was used for allowing inde pendent variations of filling ratio $y = h/D$, where h is the water depth, and approach flow velocity V in the upstream branches.

The utilization of removable plates at the first and final cross-sections of the upstream branches allowed for varying the altitudes of the branch inlets and outlets without any bottom slope modification. In this way, the tops of the straight and lateral branches were always placed at the same elevation of the downstream branch top. If one or both upstream branch diameters D_o and D_L were different from the downstream one D_u, then an offset d was present. Figure 3 shows one of the tested set-ups in which an offset at the straight inlet $d_o = D_U - D_o$ subsisted, whereas lateral and down stream branches were characterized by same diameters so that the lateral offset d_L was equal to zero.

2.2 Test program

In total, more than 150 tests were carried out. The experimental runs covered a wide range of geometrical and hydraulic parameters. The governing parameters are:

1. the diameters of the straight and lateral branches, D_o and D_L
2. the filling ratios imposed in the two upstream branches, y_o and y_L
3. the Froude numbers of the approach flows, F_o and F_L.

The range of variation of the above-mentioned parameters is reported in theTable 1.The combinations of the upstream diameters resulted in eight differ

ent set-ups. The remaining arrangement with both the Table 1. Basic geometric and hydraulic parameters varied during the performance of the experimental tests. Parameter Range of variation Unity of measure D_o 0.123; 0.190; 0.240 [m] D_L 0.123; 0.190; 0.240 [m] d_o 0.000; 0.050; 0.117 [m] d_L 0.000; 0.050; 0.117 [m] y_o 0.20; 0.30; 0.40; 0.60; 0.70 [–] y_L 0.40; 0.60; 0.70 [–] F_o 1.23–9.49 [–] F_L 1.40–7.96 [–] upstream diameters equal to 0.240 m was excluded because no offsets would exist. For each geometrical set-up, two or three different filling ratios were alternatively established in the upstream branches, depending on the specific value of the branch diameters. Fixed the objective configuration to be tested, an average number of six test-runs was performed varying the approach Froude numbers within the range specified in the Table 1. Obviously, another geometrical feature able to influence the flow pattern is the bottom offset at the straight and the lateral manhole inlets, d_o and d_L respectively. The height of the offset is correlated to the values of branch diameters, being alternatively equal to 20% and 50% of D_u (Table 1). The present study was specifically focused on the hydraulic condition of both approach supercritical flows. Froude numbers were varied being careful to avoid unstable conditions of weakly supercritical flows ($1.00 < F < 1.20$).The change of Froude number F was related to an implied variation of the corresponding discharge Q, according to following equation valid for circular pipes (Hager 2010): where g is the gravity acceleration. The equation 1 is characterized by an accuracy rate of ±5% for filling ratios between 0.20 and 0.96 (Del Giudice & Hager 2001) and the tested filling ratios are included within this range (Table 1). 2.3 Measurements The approach discharges were measured singularly by electromagnetic flowmeters with a full scale accuracy (FS) of 0.5%. Water depths and heights of shock waves in the junction were detected with a point gauge. The water depth measurement accuracy was equal to ±0.001–0.002 m due to the flow turbulence and the presence of the air water two-phase flow (Del Giudice & Hager 2001). On the other hand, the shock wave height measurement was less accurate (±0.005 m) because of the relevant oscillation of the highest level of the shock wave with large Froude numbers. Furthermore, the position of shock wave maximum was

recorded thanks to measuring strips applied on the

sidewalls of the junction manhole. Water depths in the

upstream and downstream branches were measured by

using the open windows on the top of the pipes.

3 EXPERIMENTAL RESULTS

3.1 Flow region

The diagram of state is typically used to predict the

flow types to be expected inside sewer manholes where

two-phase flows may occur. In general, for free-surface

pipe flows the specific force S acting at a generic cross

section is

The specific force represents the sum of the static (p s A)

and the dynamic (ρQV) force components, divided by

the specific weight. Here, p s is cross-sectional pressure acting at the cross-section centroid, A is the cross sectional area and ρ is the flow density.The static force component can be easily neglected when supercritical flows are considered. Accordingly, the dynamic force component, non-dimensionally almost equal to $(yF)^2$, is significant for supercritical flows (Del Giudice & Hager 2001). Therefore, Del Giudice & Hager (2001) synthetized all the junction flow patterns in a diagram of state depending on the governing parameters $(y_o \cdot F_o)$ and $(y_L \cdot F_L \cdot \cos\delta)$.This diagram was valid for 45∘ junction manhole with rounded outer side wall, equal branch diameters and almost horizontal manhole. Seven flow regions (Fig. 4) were delimited in accordance with the corresponding physical observations. Among others, flow region (4) identifies the junction flow behaviour dictated by approach supercritical flows. Since then, later studies regarding junction manholes have used the same diagram of state to describe own experimental flow patterns.

Differently from what observed by Del Giudice & Hager (2001), the present study was characterized by the variability of the upstream branch diameters. The diameter ratios $\beta_o = D_o/D_u$ and $\beta_L = D_L/D_u$ were thus introduced as additional flow regions governing parameters. In Figure 4 the parameters $(y_o \cdot F_o \cdot \beta_o)$ and

($y_L \cdot F_L \cdot \beta_L \cdot \cos\delta$), as derived from the water depths measurements at the junction manhole inlets, are plotted on the diagram of state defined by Del Giudice & Hager (2001). As shown, the experimental data points are not entirely included in flow region (4) (shaded area in Figure 4), despite only both approach supercritical flow conditions were physically considered. The junction flow behaviour was thus affected by the presence of the bottom offsets and static force components of the approach flows might be not negligible in such conditions.

According to Fig. 4, following considerations can be inferred:

– A not negligible number of points are located in flow region (3). This area was basically corresponded to approach straight and lateral flows with Froude numbers less than 2.00. It was thus defined as a transitional hydraulic condition region (Del Giudice & Hager 2001). Conversely, the abovementioned points are related to test-runs with stably supercritical approach flows. The only exception is the group of data referred to the set-up $D_o = 0.240$ m and $D_L = 0.123$ (light circles in the lower part of the region), with approach Froude numbers less than 2.00. Therefore, according to the present physical model tests, flow region (3) should be made smaller by lowering its

Figure 4. Experimental data points plotted on the diagram of state defined by Del Giudice & Hager (2001) for 45 ∘ junction manholes: (1) Approach subcritical flows, (2) Straight branch supercritical and lateral branch subcritical flows, (3) Transition to supercritical flows in both upstream branches, (4)Approach supercritical flows, (5) Choking of either lateral or straight branches, (6) Straight branch subcritical and lateral branch supercritical flows and (7) Choking of downstream branch.

upper boundary limit – Some experimental points are included in flow region (5), matched by Del Giudice & Hager (2001) to the choking condition of either the lateral or both upstream branches. In particular, most of these data points are referred to the set-up with minimum diameter for the straight branch (D_o = 123 mm). In such test-runs, the branches choking was not observed. Evidently, the existence of the bottom offset limited the occurrence of the hydraulic jump in the upstream branches, working as a sort of hydraulic discontinuity – Few data points (all of them associated to the setup with minimum diameter for the lateral branch D_L = 123 mm) fall in flow region (1). This area was defined by Del Giudice & Hager (2001) for subcritical flows in both upstream branches. Oppositely, this condition was not observed in the present investigation. However, the subcritical flow condition in both upstream branches was not carried out by Del Giudice & Hager (2001) and for this reason the domain of flow region (1) might be needed to be made smaller – Only two data points are in flow region (7). This area was corresponded to the occurrence of choking flow in the downstream branch due to the formation of a swell at the outlet cross-section. This failure was not observed in the experimental tests related to the considered data points.Therefore, in accordance with the present investigation the lower limit of the choking domain of the downstream branch should be raised or deleted if upstream branch diameters are variable and inlet offsets are present.

3.2 Shock waves

For 45 $\circ$ junction manholes with equal branch diameters, Del Giudice & Hager (2001) identified the occurrence of three shock waves: wave A, wave B and wave C. The same hydraulic features were physically described by Saldarriaga et al. (2012) for supercritical flows merging in a circular junction manhole with small drops at the manhole inlets.

3.2.1 Wave A

Shock wave A, as denominated originally by Schwalt & Hager (1995), appears when the straight branch is hydraulically dominant. This wave is orig inated by the straight inflow expansion on the inner wall of the lateral branch. Schwalt & Hager (1995) observed it only for straight flow condition, making this wave as a less interesting feature. Del Giudice & Hager (2001) observed wave A only for small lat eral filling ratio $y_L = 0.20$ and when both supercritical flow entered the junction manhole. Saldarriaga et al. (2012) stated that lateral discharge should be smaller than 10% of straight inflow so that wave A may happen.

The present study confirmed wave A as non dominant flow feature in the junction manhole. Dif ferently from Schwalt & Hager (1995), wave A was observed even for lateral inflow but its height was still not relevant compared with others shock waves. The measurement of wave A height was not simple, because of its high turbulence and thinness. In the present investigation, one of the following conditions had to be valid to determine wave A occurrence:

– Straight branch diameter ($D_o = 0.190$ m or $D_o =$

0.240 m) larger than the lateral one (D_L = 0.123 m).

For these set-ups, wave A formation was observed frequently when y_L = 0.4 and sporadically when y_L = 0.6 within Froude numbers $F_L \leq 2.50$. In the first case, when F_L was larger than about 4.50, the straight inflow passed under the lateral approach flow without any evident interaction with the flow issued by the lateral branch (Fig. 5a). The flow impinged on the opposite wall but the originated wave height was less relevant than the other one on the left sidewall.

– $D_o = D_L$ = 0.123 m and y_o = 0.60. For this geometry (Fig. 5b), if the lateral Froude number F_L was larger than 5.00, then the lateral inflow was still able to overstep the straight free-jet. Consequently, the straight flow could collide with the opposite wall.

By the way, the corresponding wave A height was smaller than in the previous scenario (Fig. 5b). Figure 5. Upstream view of supercritical junction in operation, showing shock waves A (highlighted by arrows) and B: (a) D_o = 0.190 m, y_o = 0.40, F_o = 4.62, D_L = 0.123 m y_L = 0.40, F_L = 6.14 and (b) D_o = 0.123 m, y_o = 0.60, F_o = 1.64, D_L = 0.123 m y_L = 0.40, F_L = 5.44. 3.2.2 Wave B Shock wave B originates from the collision of the lateral approach flow on the opposite manhole sidewall. This wave is typically visible for either both supercritical flows in upstream branches or only for supercritical lateral inflow alone (Schwalt 1996). If straight inflow is present, then it disturbs the impact of the lateral flow on the opposite wall with a possible reduction of the shock wave height. Otherwise, the hit of the straight approach flow on the standing shock wave B can provoke a hydraulic jump. As pointed out by many authors, shock wave B represents the main hydraulic feature to be considered in the junction manhole design. Schwalt & Hager (1995) defined shock wave B as the highest wave for supercritical junctions. Height, thickness and position of shock wave B depend strictly on the lateral inflow conditions. The effect of junction angle δ on wave B features is moderate. For this reason, junction angle δ was excluded from the group of governing parameters (Schwalt & Hager 1995). Del Giudice & Hager (2001) noticed that maximum wave B height and location can be correlated only to lateral Froude number F_L. This functional relationship was confirmed by Gisonni & Hager (2002b) for 90° supercritical junctions. The present study showed shock wave B formation as one of the main failures for junction. This shock wave (Fig. 6a) was observed by testing all the manhole set-ups for of all the considered upstream filling ratios. In some test-runs, the height of wave B resulted to be larger than the manhole bench and, consequently, a non-negligible amount of water fell out the model (Fig. 6b). Moreover, when the shock wave B occurred near the far end of the junction manhole, the flow impinged on the manhole endwall (Fig. 7a) producing a massive spray development. This poor flow condition was systematically identified for $F_L > 5.00$, independently on straight branch Froude number. As a consequence of the wave B occurrence, the outflow was shocked (Fig. 7b) and the free-surface along the downstream branch was characterized by typical cross waves with successive maxima and minima wave heights.

Figure 6. Shock wave B: (a) up view of the collision of

the flow against the manhole sidewall and formation of the

wall wave ($D_o = 0.240$ m, $y_o = 0.30$, $F_o = 4.24$, $D_L = 0.190$ m

$y_L = 0.40$, $F_L = 7.91$) and (b) detail of the overflow due to

excessive height of shock wave B ($D_o = 0.123$ m, $y_o = 0.40$,

$F_o = 6.12$, $D_L = 0.123$ m $y_L = 0.40$, $F_L = 6.61$).

Figure 7. (a) Impact of shock wave B on the manhole

endwall ($D_o = 0.240$ m, $y_o = 0.40$, $F_o = 3.73$, $D_L = 0.123$ m

$y_L = 0.60$, $F_L = 3.85$) and (b) downstream view of the

shocked flow outing from the manhole ($D_o = 0.123$ m,

y_o = 0.40, F_o = 2.46, D_L = 0.240 m y L = 0.40, F_L = 5.23).

Del Giudice & Hager (2001) observed that maxi

mum height of shock wave B could be expected to be

placed indicatively at $X_B = (x_B - x_J)/D_u = 2.00$, being

x_B and x_J the streamwise coordinates of the wave

maximum and junction point J (Fig. 1), respectively.

This indication was further confirmed by Gisonni &

Hager (2002b) for 90 ∘ junctions. According to the

present experimental data, this assumption resulted

to be roughly valid when $D_o \geq D_L$. For this set-up,

the straight inflow tended to move the shock wave

B slightly downstream of the undisturbed collision

point on the opposite sidewall (Fig. 8a). Conversely,

for lateral bottom offsets smaller than straight ones,

the straight inflow offered a weak perturbation and the

approach lateral flow could get a free impact on the

opposite sidewall (Fig. 8b).

3.2.3 Wave C

According to Schwalt (1996), when two supercritical

flows merge, a shock wave occurs near to the junction

point J. If junction manholes are considered, then the

junction shock wave is typically denominated as shock

wave C. The approach lateral flow is deviated by the

presence of the straight inflow, similarly to an abrupt wall deflection. When the junction angle is $\delta = 90^\circ$,

wave C represents the terminal portion of a shock wave

generated along the lateral conduit and due to the wall

curvature (Gisonni & Hager 2002b). Figure 8. (a) maximum wave B observed along the straight extension of the manhole ($D_o = 0.240$ m, $y_o = 0.20$, $F_o = 3.61$, $D_L = 0.123$ m $y_L = 0.40$, $F_L = 6.08$) and (b) maximum wave B located upstream of the straight extension of the manhole ($D_o = 0.123$ m, $y_o = 0.40$, $F_o = 4.23$, $D_L = 0.240$ m $y_L = 0.40$, $F_L = 3.14$). The hydraulic features of the approach supercritical flows and the junction angle δ dictate shape, magnitude and position of shock wave C. Schwalt (1996) stated that wave C might be flat-crested for small approach Froude numbers. Oppositely, large values of F_o and F_L may generate sharp-crested and air entrained shock waves C. For 45° junction manholes, the angle of the wave crest depends on the dominant flow features. If the lateral flow is dominant, then the direction of the wave should be orientated toward the sidewall in front of to the lateral entrance. For junctions hydraulically governed by the straight flow, the wave tends to be parallel to the manhole axis. The height of wave C was correlated to the lateral Froude number F_L, as for wave B. For this reason, shock waves B and C are easily comparable. Previous experimental investigations (Schwalt 1996, Del Giudice & Hager 2001) showed that wave C maximum was smaller than the highest level reached by wave B. Furthermore, the wave observed along the outer wall of the lateral branch for 90° junctions is even smaller than 45° wave C. Shock wave C was thus considered as a minor interest flow feature in the junction manhole design. Del Giudice & Hager (2001) observed wave C occurrence either for both supercritical flows in upstream branches or when straight incoming flow was supercritical and minimum condition established in the lateral branch. Saldarriaga et al. (2012) related wave C formation to the ratio Q_L/Q_o. If Q_L/Q_o is larger than 0.10, then wave C can be observed. In the present study, the presence of the bottom offsets at the upstream inlets established different occurrence conditions for wave C. In particular: – when maximum bottom offset ($d = 0.48 \cdot D_u$) was put at both upstream inlets, shock wave C was rarely observed. In most of the test-runs, the straight flow overstepped the lateral inflow (Fig. 9), without any lateral flow deflection. This circumstance was clearly identified for large straight Froude numbers ($F_o > 3.00$). This limit value increased up to 4.00, approximately, when D_L was equal to 0.190 m and 0.240 m. For smaller values of

F_o, the straight flow plunged on the lateral inflow determining a high turbulence flow area. A shock front could be identified

Figure 9. Top view of the junction flow characterized by the absence of the shock wave C (D_o = 0.123 m, y_o = 0.40, F_o = 4.78, D_L = 0.123 m y_L = 0.40, F_L = 4.93).

Figure 10. Shock front C (highlighted by the arrow) originated at the junction point (D_o = 0.123 m, y_o = 0.60, F_o = 2.98, D_L = 0.240 m y_L = 0.40, F_L = 3.43).

(Fig. 10), whose height was definitely smaller than the maximum reached by shock wave B;

– in the other set-ups, the lateral inflow impacted the straight flow. This flow collision originated a con tinuous front shock ending with the impact on the opposite sidewall (Fig. 11). In the most of the cases, it was not possible to measure separately the two flow features because of the extensive spray devel opment and the air entrainment. In few cases, the lateral flow deflection appeared to be clear (Fig. 12) and the consequent shock wave resulted to be the manhole design flow feature given its height.

4 CONCLUSIONS

Supercritical flows combining in 45° junction manhole were analyzed based on physical model investigations. The experimental facility aimed to reproduce the fre quent sewer manhole arrangement in which upand downstream branches tops are placed at the same ele

vation and branch diameters differ. This configuration Figure 11. Upstream view of shock front C (D_o = 0.240 m, y_o = 0.30, F_o = 3.74, D_L = 0.190 m y_L = 0.60, F_L = 2.97). Figure 12. Upstream view of shock front C (D_o = 0.240 m, y_o = 0.20, F_o = 4.44, D_L = 0.123 m y_L = 0.40, F_L = 2.90). implies bottom offsets at manhole inlets, whose effect on the junction behaviour had to be defined. First outcomes of the experimental campaign were herein discussed. A detailed description of the observed flow conditions was reported, with particular care to the hydraulic features governing the junction manhole design. First, the experimental observations were overlapped on the diagram of state defined by Del Giudice & Hager (2001) for 45 ∘ junction manholes with constant branch diameters and bottoms at same invert. The diameter ratios $\beta_o = D_o/D_u$ and $\beta_L = D_L/D_u$ had to be introduced among the flow region governing parameters to keep into account the branch diameters variability. Not all the experimental data are included in flow region (4) devoted to both approach supercritical flows, even though only tests with supercritical flows entering the junction manhole were herein considered. In particular, flow regions (3), (5) and (7)

(Del Giudice & Hager 2001) are not totally consistent

with the present experimental observations. For equal

approach flow conditions, the bottom offsets at the

manhole inlets inhibited the occurrence of hydraulic

jump in the upstream branches or the choking of the

downstream branch. Further analyses should clarify

the possibility to insert the bottom offset d among the

parameters governing the flow regions.

The main features of waves A, B, and C were

described including wave maximum and location.

Shock wave B resulted to be the highest wave for

supercritical junctions. In certain test-runs, the wave height exceeded the manhole bench height. Furthermore, when lateral Froude number was larger than 5.00, the wave impinged on the manhole endwall, provoking massive spray development. Waves A and C were less interesting flow features in the junction manhole design. Their maximum heights were smaller than shock wave B. The occurrence conditions of these waves are different from the previous junction researches with no bottom offsets at manhole inlets. In particular, when maximum bottom offset was put at straight inlet, shock wave C was rarely observed. Further analyses on the collected data will be carried out to establish valid expressions for computing wave maxima and locations and to define the hydraulic capacity of supercritical junction manhole in such configuration.

Del Giudice, G. & Hager, W.H. 2001. Supercritical flow in 45 ∘ junction manhole. Journal of Irrigation and Drainage Engineering 127(2): 100–108. Gisonni, C. & Hager, W.H. 2002a. Supercritical flow in manholes with a bend extension. Experiments in Fluids 32(3): 357–365. Gisonni, C. & Hager, W.H. 2002b. Supercritical flow in the 90 ∘ junction manhole. Urban Water 4: 363–372. Gisonni, C. & Hager, W.H. 2012. Idraulica dei sistemi fognari, dalla teoria alla pratica. Milan: Springer-Verlag. Hager, W.H. 2010. Wastewater Hydraulics – Theory and Practice, 2nd edition. Berlin: Springer-Verlag. Idel'cik, I.E. 1986. Handbook of hydraulic resistance. Washington, DC: Hemisphere Publishing Corporation. Oka, K. & Ito, H. 2005. Energy losses at tees with large area ratios. Journal of Fluids Engineering 127(1): 110–116. Pfister, M. and Gisonni, C. 2014. Head losses in junction manholes for free surface flows in circular conduits. Journal of Hydraulic Engineering 140(9): 06014015. Saldarriaga, J., Bermúdez, N. & Rubio, D.P. 2012. Hydraulic behaviour of junction manholes under supercritical flow conditions. Journal of Hydraulic Research 50(6): 631– 636. Schwalt, M. & Hager, W.H. 1992. Die Strahlbox. Schweizer Ingenieur und Architekt 110(27-28): 547–549 (in German). Schwalt, M. & Hager, W.H. 1995. Experiments to supercritical junction flow. Experiments in Fluids 18: 429–437. Schwalt, M. 1996. Vereinigung schiessender Abflüsse. Gwf Wasser-Abwasser, 137(6): 326–330 (in German). Zhao, C.H., Zhu, D.Z. & Rajaratnam N. 2004. Supercritical sewer flows at a combining junction: A model study of the Edworthy trunk junction, Calgary, Alberta. Journal of Environmental Engineering Science 3: 343–353. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Investigation of the hydrodynamic pressures on lock gates

during earthquakes

L. Buldgen & P. Rigo

ANAST, University of Liège, Liège, Belgium

H. Le Sourne

GeM Institute, UMR CNRS 6183, ICAM Nantes, France

ABSTRACT: In seismic areas, the design of lock gates has to be done by accounting for the total hydrodynamic

pressure generated by an earthquake. However, doing so is not straightforward because it is required to account

for the fluid-structure interaction: the water pressure has an influence on the gate vibrations and vice versa. A

solution to circumvent this difficulty is to perform finite element simulations, in which the fluid and the gate

have to be modeled. With the recent developments of computational fluid dynamics, it is possible to get accurate

numerical results. Unfortunately, such an approach is often time demanding and not always well-suited for lock

gate design. Consequently, the aim of this paper is to

present a meshless method that allows a rapid prediction of the total hydrodynamic pressure applied on a plane lock gate during earthquakes. These analytical developments are then compared with numerical simulations.

1 INTRODUCTION

In seismic areas such as Central America, the Middle East of the west coast of Latin America for example, the potential ground accelerations during an earth quake may generate non-negligible water pressures that have to be taken into account in the design of lock gates. However, doing so is a real challenge because it requires to correctly model the fluid-structure interac tion: the pressure influences the gate vibrations, which in turn has an effect on the pressure field.

As a consequence, there is a coupling between the fluid and the structure that can be considered numerically by using computational fluid dynamics for example. However, such an approach is not well suited for lock gates, because the dimensions of the lock chamber are such that modeling entirely the fluid domain often leads to prohibitive meshing and cal culation efforts. Therefore, other solutions have to be investigated.

One option is the quite popular added-mass method. In this approach, the Westergaard (1933) or Housner (1957) equations are used to derive lumped masses that are distributed on the structure to represent the

hydrodynamic pressure effect. However, as illustrated

by Buldgen et al. (2015), doing so may leads to non

conservative results, particularly in the case of quite

flexible gates.

In order to circumvent the modeling efforts required

to perform finite element simulations and to avoid the

approximations of the added-mass method, another

option consists in developing analytical solutions to get a quite reasonable estimation of the hydrodynamic pressure. This is precisely the aim of research work presented in this paper, in which we consider a basic lock chamber bounded by two identical downstream and upstream gates, having a height H and a width " (Figure 1). The length of the lock chamber and the water level are denoted by L and h s respectively. During an earthquake, this configuration is submitted to three different seismic acceleration components respectively applied along the x, y and z axes. However, the effect of the vertical and transversal contributions will not be considered here, so only the longitudinal acceleration applied along the x axis will be investigated. This one is denoted by $\ddot{X}(t)$ and is assumed to exhibit the time evolution represented in Figure 1 for example. The purpose of the present work is now to derive the total hydrodynamic pressure applied on the gates during the seism. 2 DERIVATION OF THE WATER PRESSURE When the lock structure is submitted to the seismic ground acceleration $\ddot{X}(t)$, an additional hydrodynamic pressure is applied on the gates. This phenomenon is due to the fact that the water confined between the gates is also accelerated and therefore exerts additional forces on the containing structures. According to Abramson (1966), the resulting total pressure has three different contributions:

Figure 1. Three dimensional view of a lock chamber.

1) The rigid impulsive part, which is derived under

the assumption of perfectly rigid gates. With this

hypothesis, the structure is supposed to move in unison with the ground without exhibiting any vibration.This rigid-body motion results in a water pressure p r that is said to be rigid impulsive.

2) The rigid flexible part, which comes from the fact that the gates are not ideal rigid structures. Their own vibrations thus superimpose to the base ground accelerations and this movement produces an additional water pressure p f that is said to be flexible impulsive.

3) The convective part, which comes from the slosh ing appearing at the free surface of the lock chamber. This wave motion is responsible for a water pressure p c that is said to be convective.

2.1 Hypotheses and basic equations

In order to find analytical solutions for the p c , p r and p f , some additional assumptions are required to solve the Navier-Stokes equations that govern the fluid motion. To do so, in addition to the hypothe ses of an inviscid fluid, the assumption is made of an incompressible and irrotational flow, i.e.:

whereV is the fluid velocity vector. With the previous hypotheses, it is shown by Currie (2003) for example that Navier-Stokes equations may be simplified in the following way: which is known as the Laplace equation and where φ is a velocity potential such that: p being the water pressure and ρ the fluid mass density. Although they are quite restrictive, the previous hypotheses seems however reasonable in the case of a lock structure, where the fluid flow during an earthquake may be reasonably supposed to satisfy the previous conditions. Furthermore, they are widely used in the literature and international standards dealing with the seismic design of containers.

2.2 The rigid impulsive pressure The rigid impulsive pressure p_r may be calculated by solving equation (2), with the following boundary conditions (Figure 1): Equations (4a), (4c) and (4d) simply state that the fluid has to follow the displacements imposed at the boundaries of the lock chamber. Equation (4b), describes the free surface condition.

Figure 2. Total displacement at the lock gates.

Considering the abvove boundary conditions and

accelerogram, it is possible to find an analytical solu

tion to equation (2), which has been achieved by

Abramson (1966), Epstein (1976), Haroun (1984) or

Housner (1957) amongst others. Considering these

developments, the rigid impulsive pressure p_r on the

gate may be written as follows:

where $\beta_n = (2n - 1)\pi / L$ and $\ddot{X}(t)$ is the ground accel

eration.

2.3 The flexible impulsive pressure

The pressure given by equation (5) is not the total

pressure acting on the gate. Indeed, p_r is calculated

under the assumption that the gates are perfectly rigid

and therefore exactly follow the ground accelerations.

However, as depicted in Figure 2, the total motion at

the gates is given by:

where the first contribution $\ddot{X}(t)$ is due to the earth quake, while u(t) is associated to the gate deformations

as a flexible structure.

With the previous considerations, the boundary

condition given by equation (4d) is not sufficient and

should be replaced by:

Solving equation (2) with the above boundary condi

tions (4a), (4b), (4c) and (4e) allows to determine the flexible impulsive pressure p f . This has been achieved by Kim et al. (1996) who showed that: where: with " m = " if m = 0 and " m = "/2 if m > 0. Finally, because of the linearity of the boundary conditions (4) and of the Laplace equation (2), the total impulsive pressure p satisfying the simultaneously (4d) and (4e) may be found simply by adding p r and p f to get: As a consequence, it is worth mentioning that equation (9) is obtained by considering the total displacement U (t) given by equation (6). This means that the fluid-structure interaction is correctly captured by this solution. 2.4 The convective pressure The derivation of the convective pressure may be done by solving equation (2) with the boundary conditions (4a), (4c) and (4d), but equation (4b) is not valid anymore. Indeed, if one aims to account for the sloshing in the lock chamber, it is required to write that the pressure has to be equal to zero at the boundary of the deformed free surface. In other words, considering y = h s in equation (4b) is no longer valid. A detailed analytical derivation of p c is provided by Ibrahim (2005) or Graham & Rodriguez (1952) amongst others. In fact, the convective part is not of primary interest for the seismic analysis of lock gates. Indeed, it may be shown that the frequencies of the sloshing modes are given by: where g is the acceleration of gravity. In the case of a lock chamber with a length L equal to 100 m and a water level h s of 10 m, the frequency of the fundamental sloshing mode (n = 1) is more or less equal to 0.05 Hz, which is clearly outside the main frequency range of seismic accelerograms. Consequently, this phenomenon is not considered here. This is even more evident if we consider the height of the sloshing wave. For the previous values of L and h s , the theoretical

Figure 3. Description of the lock gate.

evaluation of the sloshing amplitude does not exceed

10 cm, which only represents 1% of the water depth.

As a consequence, the previous considerations

clearly show that p_c can be neglected with respect to

the other contributions p_r and p_f.

3 DYNAMIC ANALYSIS OF THE GATE

3.1 Fluid-structure interaction

The fluid-structure interaction is entirely captured by

equation (7), where it is clearly shown that the flexible

pressure p_f is related to the gate displacements u(t).

The main difficulty is now to calculate u(t), which

requires to write the equilibrium equation of the gate

submitted to an external pressure p.

In the present situation, we consider a plane lock

gate that is made of a plating reinforced by horizontal

and vertical stiffeners having all a T-shaped cross

section (Figure 3). The gate is supposed to be simply

supported at the lock walls (i.e. along AB and CD) and

at the bottom (i.e. along BC) if a sill is present. For this

type of structure submitted to an external pressure p,

assuming that the out-of-plane displacement u(t) are

predominant, it is shown by Buldgen (2014) that the

dynamic equilibrium equation writes:

wherep s is the structural mass density, f_d are the damp

ing forces, D is the plating flexural rigidity, $f_{h,n}$ are
the

forces transmitted by the horizontal stiffeners and $f_{v,n}$

are the forces transmitted by the vertical stiffeners.The detailed analytical expressions of these expressions are given by Buldgen (2014). Ideally, equations (7) and (11) should be solved simultaneously to get the displacement u(t) and the pressure p. However, finding an analytical solution to this problem is almost impossible. For this reason, another approach is used to deal with the fluidstructure interaction. 3.2 Modal decomposition The modal decomposition consist in writing u(t) as a linear combination of the gate eigenmodes ψ j (y, z), i.e.: where N is the number of modes used in the decomposition and q j (t) are unknown time coefficients that have to be determined. As detailed by Buldgen et al. (2015), the eigenmodesψ j (y, z) may be calculated analytically by using the Rayleigh-Ritz method. Another approach is to perform a modal analysis of the gate to get a numerical evaluation of the mode shapes. As detailed by Buldgen et al. (2015), these two solutions are practically equivalent, in the sense that the analytical derivation of the eigenmodes provides a reasonable estimation of the numerical results. In the optic of applying the virtual work principle, a virtual displacement field δu(t) is also defined in accordance with equation (12): where δq k (t) are the virtual time coefficients. Rewriting the equilibrium equation (11) in the following way: with: it is possible to project this equation in the basis of the eigenmodes: As a final step, introducing the modal decompositions (12) and (13) of u(t) and δu(t) in equation (15) and

assuming a Rayleigh-type damping for the forces f d ,

we get the following matrix equation:

where q(t) is a vector containing the coefficients q j (t)

of equation (12), δq T (t) is the transpose of the vec

tor containing the coefficients δq k (t) of equation (13),

α and β are the Rayleigh damping coefficients and

[M], [J], [K] are (N × N) matrices that can be calcu

lated analytically according to the formulae given by

Buldgen (2014).

In fact, writing equations (15) or (16) is nothing else than expressing the virtual work principle. This theorem states that equating the internal and external virtual works for any kinematically admissible dis placement field leads to a necessary and sufficient equilibrium condition for the structure. In equation (15), the left-hand side represents the internal virtual energy, while the right-hand side corresponds to the virtual work performed by the external forces (i.e. the inertia, damping and pressure forces). As the global equilibrium is guarantee for any admissible displace ment field $\delta u(t)$, equation (16) has to be satisfied for any vector $\delta q^T(t)$, which leads to:

This last relation is nothing else than a set of N differential equations that allows to determine the unknown time coefficients $q_j(t)$ for a given accelero gram $\ddot{X}(t)$. In fact, (17) is very similar to the dynamic equilibrium equation of a structure submitted to an external pressure force $P\ddot{X}(t)$. The only difference is due to the matrix [J] that has to be added to the struc tural mass matrix [M]. In fact, $[J]\ddot{q}$ represents the flexible impulsive pressure p_f ($P\ddot{X}(t)$ corresponding to the rigid impulsive pressure p_r), which means that the fluid-structure interaction is entirely captured by the matrix [J].

As a final comment, it should be mentioned that due to the orthogonality of the eigenmodes ψ j (y, z),

the stiffness and damping matrices [K] and [M] are

diagonal. However, this is not the case for [J], which

means that the N equations given by (17) cannot be

decoupled. This set of equations may be solved by the

Newmark integration scheme for example.

4 NUMERICAL COMPARISON

In order to corroborate the (semi-)analytical develop

ments of the previous sections, we can compare them

with finite element analyses. The goal of this section

is to check if the procedure detailed here above is able

to give reasonable approximation of the total hydro

dynamic pressure induced on a lock gate during a

seism. Figure 4. Numerical model of the upstream and downstream gates. Table 1. Properties of the reinforcing system (m). Web Web Flange Flange height thickness width thickness Horizontal girders 0.98 0.02 0.4 0.025 Vertical frames 0.98 0.02 0.5 0.025 Horizontal 0.21 0.006 0 0 stiffeners 4.1 Description of the structure For this comparison, we consider a lock chamber with a total length L of 50 m and filled with water up to a level h s of 8 m.The upstream and downstream gates are identical and depicted in Figure 4. The plating is modeled with Belytschko-Tsay shell elements. To reduce a little bit the size of the model, the reinforcing system is not explicitly represented with shell elements, but rather with Hughes-Liu beams. More details about the formulation of these elements are provided by Hallquist (2006). This structure has a square plating, with H = "= 13.1 m and a thickness t p of 1.2 cm. It is reinforced by six vertical frames and five horizontal girders. The first ones are regularly placed over the width ", with a spacing of 2.62 m. The disposition of the girders is not regular, as the reinforcement is more important near the bottom of the gate. Some smaller horizontal stiffeners are also present. All these elements have a T-shaped cross-section. The geometrical and material properties are summarized in Table 1 and Table 2. The mesh of the gates is quite coarse, with a more or less regular size of 20 × 20 cm for the shell elements. This choice is due to the necessity of limiting the

Table 2. Material properties.

Mass density ρ_s (kg/m^3)	7850
Young modulus E (GPa)	210
Poisson ration ν (-)	0.3

Figure 5. Longitudinal component of the seismic accelera tion.

size of the model. Nevertheless, simulations on more refined models were also performed, with a meshing of 5 × 5 cm or 10 × 10 cm. However, it was observed that the obtained results were not too different from the present ones.

4.2 Description of the fluid

Numerical analyses are performed using the software ls-dyna. The fluid is modeled as an elastic medium, with constant stress solid elements affected by a fluid material law without shearing. This particular law is called mat_elastic_fluid in the ls-dyna manual of Hallquist (2006). In fact, the fluid is seen as an elastic medium with a mass density ρ and a bulk modulus K, for which the stress and strain rates are related by:

$[\sigma]$ and $[\varepsilon]$ being respectively the stress and strain ten sors. Equation (18) shows that there is no shearing in this material and that the pressure rate is directly proportional to the change of volume. This is consis

tent to model the behavior of an inviscid fluid in small

displacements (no turbulence), which should be appro

priate for seismic analyses. Consequently, this material

law has been chosen with the parameters ρ and K equal

to 1000 kg/m 3 and 2.25 GPa respectively. The mesh of

the fluid domain is regular, with an approximate size

of 19 × 19 × 19 cm.

It is worth mentionning that the fluid domain and

the lock gates do not share any node in common, which

means that the fluid nodes are distinct from the solid

ones. The ls-dyna penalty contact algorithm is then

used to simulate the interaction between the plate and

the surrounding liquid. This allows the fluid to slide on
Figure 6. Fourier transform of the seismic acceleration.
the flexible walls without friction, but prevent it from passing through the structure. 4.3 Description of the seismic action Boundary conditions are as described in section 1 and supports AB, BC and CD (Figure 3) are submitted to the longitudinal acceleration $\ddot{X}$ of Figure 5. The Fourier transform of this signal, represented in Figure 6, shows that the main part of the seismic excitation is located in the frequency range [1 Hz; 15 Hz]. 4.4 Presentation of the results As a final comment before detailing the results of the numerical simulations, it is worth mentioning that the consistency of the previous model has been first investigated to corroborate the use of an elastic medium for the fluid domain. In this optic, the upstream and downstream gates have been both affected with a rigid material law, which forces them to move as perfectly rigid bodies. With these modifications, the pressure computed by ls-dyna should be close to the solution of the Laplace equation given by equation (2) and the boundary conditions (4a) to (4d), for which the analytical solution (5) is extensively used in the literature. Considering the acceleration profile depicted in Figure 5 for $\ddot{X}(t)$, it is found that the discrepancy

between the numerical and analytical solutions does not exceed 7%. As this observation is quite satisfactory, the finite element model detailed in the previous sections has been kept as a reference to investigate the validity of the developments made in section section 3. As a matter of validation, we can compare the semi-analytical prediction of the time evolution characterizing the total resulting hydrodynamic pressure force applied on the gate with the solution calculated by ls-dyna for the seismic acceleration of Figure 5. The results are plotted in Figure 7, from which a quite good agreement between the two curves is observed. From this picture, it transpires that both solutions sometimes tends to be out-of-phase and that the analytical force amplitude tends to be greater than the numerical one. However, this is conservative as it means that the total resulting pressure force applied on the gate is overestimated.

Figure 7. Comparison of the total resulting pressure force calculated analytically and provided by ls-dyna.

Figure 8. Simplification of the finite element model.

5 CONCLUSION

This paper presents the analytical equations of the total hydrodynamic pressure applied on a lock gate during an earthquake. This latter is seen as the sum of three different contributions that are known as the convective, rigid and flexible pressures. For each of them, analytical solutions can be found by solving the Laplace equation with the appropriate boundary conditions.

The most difficult point in this process is the deriva tion of the flexible pressure p_f, because this has to be done by accounting for the fluid-structure interaction. Consequently, an analytical solution for p_f is found by considering simultaneously the Laplace equation and the dynamic equilibrium of the lock gate. In order to find an analytical solution to this system, the gate displacements are decomposed in the modal basis. The eigenmodes of the structure can be found either by an analytical method based on the Rayleigh-Ritz method, or they can be directly extracted from a numerical modal analysis. With the modal decomposition of the gate displacements, it is possible to project the dynamic equilibrium equation of the gate in the modal basis. Applying the virtual work principle leads to a set of equations that can be solved numerically using the Newmark integration scheme. Doing so provides the time evolution of the displacements that can be used to calculate the flexible pressure. In order to validate the analytical derivation of the water pressure, comparisons are made with solutions provided by using the finite elements code ls-dyna. The agreement being quite satisfactory, the simplified methodology exposed in this paper can be used to get a rapid estimation of the time evolution of the hydrodynamic pressure applied on a lock gate. This semi-analytical method avoid performing heavy numerical simulations during the pre-design of such structures. Indeed, instead of modelling extensively

the fluid domain with finite elements (which is usually

time demanding), the seismic action may be directly

taken into account by applying equivalent nodal forces

that are predicted analytically (Figure 8).

ACKNOWLEDGMENTS

The authors would like to thank the MSC Software

Corporation for its support with the finite element soft

ware msc/patran-nastran that were used in many

steps of this research. msc patran has been utilized

as a powerful pre-processor for building the mod

els required to study the seismic response of lock

gates.mscnastran has been used to performed modal

analyses of the structures.

Abramson, H.N. 1966. The dynamic behavior of liquids in

moving containers.Washington: NationalAeronautics and

Space Administration.

Buldgen, L. 2014. Simplified analytical methods for the

crashworthiness and the seismic design of lock gates (PhD

thesis). Liège: University of Liège.

Buldgen, L. 2015. A simplified analytical method to evalu

ate the seismic pressure on plane lock gates. Engineering

Structures 100: 522–534. Currie, I.G. 2003. Fundamental mechanics of fluids. New York: Marcel Dekker Inc. Epstein, H.I. 1976. Seismic design of liquid storage tanks. Journal of the Structural Division 102: 1659–1673. Graham, E.W. & Rodriguez, A.M. 1952. The characteristics of fuel motion which affect airplane dynamics. Journal of Applied Mechanics 19: 381–388. Hallquist, J.O. 2006. LS-DYNA theoretical manual. Livermore: Livermore Software Technology Corporation. Haroun, M.A. 1984. Stress analysis of rectangular walls under seismically induced hydrodynamic loads. Bulletin of Seismological Society of America 74: 1031–1041. Housner, G.W. 1957. Dynamic pressures on accelerated fluid containers. Bulletin of Seismological Society of America 47: 15–37. Ibrahim R.A. 2005. Liquid sloshing dynamics: theory and applications, Cambridge: Cambridge University Press. Kim, J.K., Koh, H.M. & Kwahk, I.J. 1996. Dynamic response of rectangular flexible fluid containers. Journal of Engineering Mechanics 122: 807–817. PIANC Working Group 151. 2016 (to be published). Design of lock gates under seismic action. Brussels: World Association for Waterborne Transport Infrastructure. Westergaard, H.M. 1933. Water pressure on dams during earthquakes.Transactions of theAmerican Society of Civil Engineers 98: 418–433. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Stilling basin design for inlet sluice with vertical drop structure: Scale model results vs. literature formulae

J. Vercruysse & K. Verelst

Flanders Hydraulics Research, Antwerp, Belgium

T. De Mulder

Hydraulics Laboratory, Department of Civil Engineering, Ghent University, Ghent, Belgium

ABSTRACT: Within the framework of the Updated Sigmaplan, Flood control Areas (FCA) with a Controlled

Reduced Tide (CRT) are set up in several polders along the tidal section of the river Scheldt and its tributaries.

The reduced tide is introduced by means of simple inlet and outlet sluices located in the levee between the river

and the polder. In recent designs, the inlet sluice is placed on top of the outlet sluice. At water intake, the water

drops from the brink of an inlet sluice apron into a stilling basin, integrated with the floor slab of the underlying

outlet sluice. Flanders Hydraulics Research performed a scale model based review of several desktop designs of

this type of combined inlet and outlet structures. This paper compares the scale model results and predictions of

literature formulae for drop structures and stilling basins, upon which the desktop designs were based. Several

types of sluice geometries – with a straight stilling basin, a locally deepened stilling basin and a stilling basin

with baffle blocks – were studied. This comparison exercise concludes that suitable formulae are available for

a stilling basin design when the tailwater depth at the polder side equals the conjugate water depth. For higher

tailwater depths, no suitable literature formulae seem to be available and physical (or numerical) modelling is

recommended.

1 INTRODUCTION

In 1976 a major flooding event occurred in the Scheldt

Estuary (Belgium). After this event the so called

Sigmaplan was elaborated in 1977. One of the mea sures of the Sigmaplan to mitigate flood risks was to build a set of Flood Control Areas (FCA's). To this end, well-chosen polders along the tidal river Scheldt and its tributaries were selected. The FCA's are filled, dur ing storm tide, through overflow over a lowered levee and emptied, during low tide, through an outlet sluice. In this way, the top levels of the storm surge are lowered by storing a volume of water in the FCA's.

Due to industrialization and urbanization many of the estuarine habitats have disappeared or degraded over the course of time. These changing physical cir cumstances, together with new insights resulted into an update of the original Sigmaplan in 2005. Besides safety against flooding, the Updated Sigmaplan aims to contribute to the restoration of estuarine habitats. One measure is to create a semi-diurnal, Controlled Reduced Tide (CRT), in some of the FCA's. To this end, an FCA is equipped with a well-designed inlet sluice, positioned high in the levee between the river and the polder, and an outlet sluice at a lower position in the levee.

The first two designed FCA-CRT's are Lippenbroek and Kruibeke-Bazel-Rupelmonde.The inlet and outlet Figure 1. Principle drawing of a combined inlet and outlet structure. sluices in these FCA-CRT's were separated in two different structures (De Mulder et al., 2013). For the new

FCA-CRT's, the inlet and outlet sluices are combined in one structure (Vercruysse et al., 2013) of which a principle drawing is given in Figure 1. Note that the inlet sluice is put on top of the outlet sluice. At water intake, water from the river flows into the inlet sluice and is subsequently subjected to a vertical drop at the brink of the inlet sluice's floor slab. To dissipate the energy of the incoming water, a stilling basin is provided, integrated in the floor slab of the outlet sluice. Downstream of the concrete section, a bottom protection with gabions is foreseen. Combining the inand outlet sluice into one structure has economic and ecological benefits (De Mulder et al., 2013). The required sill levels and (total) cross-sectional areas of the combined inlet and outlet structures have

Figure 2. Test section of scale model. Arrow indicates the

flow direction at water intake.

been determined at Flanders Hydraulics Research by means of a numerical model study.The goal is to obtain a suitable reduced tide, i.e. aiming at a distinct spring tide/neap tide cycle, in the FCA-CRT. The detailed structural design of the FCA-CRT's, and the combined inlet and outlet structure, was outsourced by the Water way Administration to distinct consulting engineering firms. They made a desktop design based on literature formulae. Flanders Hydraulics Research performed a scale model based review of several desktop designs of this type of combined inlet and outlet structures. An overview of the scale model and the tested geometries are presented in section 2. To facilitate the comparison between scale model results and litera ture formula the results of the scale model tests are presented dimensionless, based upon the length and velocity scales described in section 3. The following sections present a comparison between literature for mulae and scale model results for the conjugate water depth (section 4), the drop length (section 5) and the end of the hydraulic jump (section 6). Most litera ture formulae are valid for a tailwater depth equal to the conjugate water depth. Section 7 discusses some examples of flow patterns and velocities for higher tailwater depths then the conjugate water depth. A conclusion is formulated in section 8.

2 SCALE MODEL TESTS

2.1 Model scale

The hydraulic review was carried out by means of scale model experiments, adopting a so-called 2DV model ing approach. The flow pattern was only studied in the vertical symmetry plane of one combined inlet and outlet structure.

The scale model of the combined inlet and outlet structure (geometrical scale 1:8 using Froude scaling), was built into a section of the current flume with a length of 15.0 m, a width of 0.56 m and a height of 1.0 m. Figure 2 shows the scale model’s test section. By using exchangeable plates fixed on movable mechan ical lifts, it was possible to test the different structures and easily adopt geometrical changes. Aeration of the

falling nappe is provided by a tube mounted under the

ceiling of the outlet culvert.

The following measurements were carried out:

• water level upstream and downstream with 2 elec

tronic water level gauges (BTL5-E17, from Balluff,

accuracy 1 mm). • discharge by electromagnetic discharge meters on the supply conduits (Aquaflux K from Khrone, accuracy better than 1% of measured discharge), • near bottom velocity with 2 electromagnetic point velocimeters (EMS from Deltares, accuracy 1% of the measured velocity). Besides these measurements, also visual registrations were carried out of: • the drop length, • the end of the hydraulic jump, • the water level in the outlet culvert. 2.2 Tested geometries In the timeframe of the research, January 2012 till December 2013, the combined inlet and outlet structures of following FCA-CRT's were tested: • Bergenmeersen (BGM), • Dijlemonding (DLM), • Vlassenbroek (VLB), • De Bunt (BNT). These combined inlet and outlet structures differ in the drop height and in design choices made by the respective consulting firms. First the initial desktop designed geometry was tested in the scale model. Based on the test results, alternative geometries were defined in agreement with the Waterway Administration and tested. Due to the long roof above the stilling basin of DLM the scale model tests for this structure were mainly focused on the flow pattern downstream of the structure rather than on the flow pattern in the stilling basin. For this reason, the results of DLM are not comparable with the results of the other tested geometries The locally deepened stilling basin for BNT is identical to a tested geometry for VLB, although the roofs of these geometries differ. Due to the identical stilling basin only a limited number of experiments were carried out for BNT. Consequently this paper only presents results of BGM and VLB. The tested geometries of BGM and VLB are named with the abovementioned abbreviation of the FCACRT (BGM, VLB), followed by "G" and a follow-up number starting with 1, being the desktop designed geometry. For the analysis in this paper the tested geometries are divided into 3 categories: • Geometries with a straight stilling basin, see Figure 3. • Geometries with a locally deepened stilling basin with end sill, see Figure 4. • Geometries with a straight or locally deepened stilling basin with baffle blocks, see Figure 5.

Note that the dimensions in these figures are expressed relative to the drop height Δz, defined as the vertical distance between the upper face of the inlet sluice apron and the upper face of the stilling basin apron. For BGM, only geometries with a straight stilling basin (without end sill) and a straight stilling basin with

Figure 3. Tested geometries with a straight stilling basin.

Arrow indicates the flow direction at water intake.

Figure 4. Tested geometries with a locally deepened stilling basin and end sill. Arrow indicates the flow direction at water intake.

Figure 5. Tested geometries with baffle blocks. Arrow indicates the flow direction at water intake.

baffle blocks were tested. To investigate the influence of the outlet sluice for BGM also a geometry with a vertical wall below the brink of the inlet culvert apron, was tested (BGM G3). For VLB geometries belonging to the 3 categories were tested.

The baffle blocks were designed according to the design rules for a USBR type III stilling basin (Peterka, 1984; Thompson & Kilgore, 2006). Note that this implies an application of the design rules outside their validity range (which corresponds to a stilling basin at the toe of a downward sloping chute, contrary to the vertical drop pertaining to the present structures). For the design of the baffle blocks, the start of the basin is defined as the drop length determined by the formula from Chanson (section 5, eq. 6). Figure 6. Schematic illustration hydraulic jump. The structures of BGM and VLB are relatively limited in length; the (concrete) stilling basin apron ends between 4.00 Δz and 5.61 Δz downstream of the drop (not indicated in Figures 3, 4 and 5). 3 DIMENSIONLESS PRESENTATION OF RESULTS At water intake, water from the river flows into the inlet sluice and is subjected to a vertical drop at the brink of the inlet sluice's floor slab. Consequently a hydraulic jump is formed. This is schematically presented in Figure 6, also indicating the symbols that are used in this paper. To facilitate the comparison of the tested geometries, the results in this paper will be presented in dimensionless form. Therefore the drop height Δz is used as length scale for the lengths and levels. The critical water depth, scaled with the drop height Δz, is used to quantify the discharge per unit width. The critical water depth is namely only a function of the discharge per unit width q [$m^3/(ms)$]: where d_c = critical water depth [m], Δz = drop height [m], q = discharge per unit width [$m^3/(ms)$], g = gravity acceleration [m/s^2]. For the dimensionless presentation of the velocity, the theoretical maximum velocity in the stilling basin V_1 will be applied as velocity scale, which is defined as the ratio of the discharge per unit width q and the water depth before the hydraulic jumpY_1 ($V_1 = q/Y_1$). The latter quantity will be computed with the formula from Rand (1955): whereY_1 = water depth before jump [m], d_c = critical water depth = $(q^2/g)^{1/3}$ [m], Δz = drop height [m]; q = discharge per unit width [$m^3/(ms)$], g = gravity acceleration [m/s^2].

Figure 7. Variation of conjugate water depth in function of

critical water depth. Geometries with a straight stilling basin.

4 CONJUGATE WATER DEPTH

To transform the supercritical flow after the drop to subcritical flow, a minimal tailwater depth, the conju gate water depth, is necessary. This section compares the measured (in the scale model) and the computed (with literature formulae) conjugate water depth.

4.1 Formulae

For calculating the conjugate water depth downstream

of a vertical drop, Y_2 , formulae from Rand (1955) and

Chanson (2002) are available.

Formula Rand (1955):

Formula Chanson (2002):

where Y_2 = conjugate water depth [m], d_c = critical

water depth = $(q^2/g)^{1/3}$ [m], �z = drop height [m],

q = discharge per unit width [$m^3/(ms)$], g = gravity

acceleration [m/s^2].

4.2 Results

4.2.1 Straight stilling basin

The measured conjugate water depths for the geome

tries with a straight stilling basin (Figure 3) are

presented in Figure 7 as a function of the critical water

depth.

Note in Figure 7 that there is no noticeable dif

ference between an aerated and a non-aerated falling

nappe and between an outlet sluice (BGM G1 andVLB

G2) and a vertical wall (BGM G3). In Figure 8 the fore

going results are compared with the conjugate water

depths according to equations (3) and (4). Figure 8. Comparison of conjugate water depth formulae with measurements. Geometries with a straight stilling basin. Table 1. Comparing trend line coefficients with formula Rand and Chanson. Rand Chanson Experiments (1955) (2002) factor 1.66 1.565 1.835 exponent 0.81 0.809 0.930 Figure 9. Conjugate water depth in function of critical water depth. Geometries with a locally deepened stilling basin. Figure 8 also contains a trend line of the measurements. This trend line is of the same power law type as equations (3) and

(4). The computed parameters (factor and exponent) are presented in Table 1. Note in Figure 8 and Table 1 that the experimental trend line is somewhat steeper than the curves corresponding to the Rand and Chanson formulae.

4.2.2 Locally deepened stilling basin

For the geometries with a locally deepened stilling basin, Figure 9 presents the variation of the conjugate water depths as a function of the critical water depth. Note that the conjugate water depth is referenced to the bottom of the locally deepened stilling basin. For VLB G5, a suitable hydraulic jump was formed independent of the tailwater depth for all tested critical water depths. Therefore, no results for VLB G5 are presented in Figure 9.

Figure 10. Conjugate water depth in function of critical water depth. Geometries with baffle blocks.

For VLB G4 and G6, a rather good agreement is found between the measurements and the formulae from Rand and Chanson. For VLB G3, however, the conjugate water depth is underestimated for higher critical water depths. During the tests it was noticed that for VLB G3 the falling nappe reached the apron downstream of the locally deepened stilling basin at higher critical water depths.

4.2.3 Baffle blocks

The measured conjugate water depths for the experiments with baffle blocks (Figure 5) are presented in Figure 10. Only results for BGM G8 and G9 are presented. For VLB G7, a suitable hydraulic jump was formed independent of the tailwater depth for all tested critical water depths. The baffle blocks for BGM were designed for a critical water depth of $d_c / \Delta z = 0.62$.

Thompson & Kilgore (2006) mentioned, although not recommended, that the conjugate water depth can be reduced with a factor 0.85 when using a USBR type III stilling basin. For this reason, the conjugate water depths computed according to formulae (3) and (4) and reduced with a factor 0.85, are presented in Figure 9.

Figure 10 shows that the baffle blocks only reduce the conjugate water depth with a factor 0.85 for the design critical water depth ($d_c/\Delta z = 0.62$). For higher (respectively lower) critical water depths the computed conjugate water depth underestimates (respectively overestimates) the measured value.

Figure 10 shows also that the measurements do not follow the same power type law as both the literature formulae.

5 DROP LENGTH

This section presents a comparison between the measured and the computed drop length.

5.1 Formulae

For calculating the drop length downstream of a vertical drop, formulae from Rand (1955) and Chanson (2002) are available. Figure 11. Visually determination of the drop length. Figure 12. Variation of drop length in function of critical water depth. Formula Rand (1955): Formula Chanson (2002): where L_D = drop length [m], d_c = critical water depth = $(q^2/g)^{1/3}$ [m], Δz = drop height [m], q = discharge per unit width [$m^3/(ms)$], g = gravity acceleration [m/s^2]. Note that the formulae above are representative for an aerated nappe. When the nappe is not aerated the drop length is smaller. 5.2 Results In the scale model the drop length was visually determined for a flow pattern with downstream supercritical flow, as illustrated in Figure 11. The measured and computed drop lengths for BGM G1, VLB G2 and VLB G6 are compared in Figure 12. The drop length was not determined for VLB G3 and G4 (respectively VLB G5) because the drop height of these geometries are equal to VLB G2 (respectively VLB G6). The drop length computed with the formula from Chanson is a good estimation of the measured drop

length for VLB G2 aerated, VLB G6 aerated and (for critical water depths exceeding $d_c/\Delta z = 0.48$) BGM G1 aerated. The drop length tends to reduce when the nappe is not aerated.

6 END OF HYDRAULIC JUMP

During the experiments, the end of the hydraulic jump was visually registrated as the location where no more turbulent bursts were visible at the free surface. The distance between this location and the vertical drop is then compared with predictions (from literature for mulae) of the cumulative value of the drop length and the length of the subsequent hydraulic jump.

6.1 Formulae

The drop length, L_D, is computed according to the formula from Chanson (eq. 6), see section 5.

The length of the hydraulic jump, L_j, is computed using the following formulae:

Rand (1955):

Silvester (1964):

Hager et al. (1990):

where L_j = length of hydraulic jump [m], Y_2 = conjugate water depth (eq. 3) [m], Y_1 = water depth before jump (eq. 2), Fr_1 = Froude number before jump = $(q/Y_1)/(g Y_1)^{1/2}$ [–], q = discharge per unit width [$m^3/(ms)$], α_j = coefficient (=22, when $4 \leq Fr_1 \leq 12$).

6.2 Results

The end of the hydraulic jump, i.e. the cumulative length of the drop and the hydraulic jump, $L_D + L_j$, is presented as a function of the critical water depth in Figures 13 and 14 for the geometries with a straight stilling basin (Figure 3) and for those with a locally deepened stilling basin (Figure 4), respectively.

For a geometry with baffle blocks no results will be presented in this section, due to the increased uncertainty of the visually determined end of the hydraulic jump.

Taking into account the visual registration of the end of the hydraulic jump, there is a rather good agreement between the measured cumulative length and the range of cumulative lengths computed with formulae of Silvester and Rand. For some combinations of geometry

Figure 13. Cumulative length of drop and hydraulic jump in function of critical water depth. Geometries with a straight stilling basin.

Figure 14. Variation of cumulative length of drop and hydraulic jump in function of critical water depth. Geometries with a locally deepened stilling basin.

and critical water depths, there is a significant increase in the cumulative length when the nappe is aerated, whereas for other combinations, there is no noticeable effect. For VLB G3, G4 and G6, a rather good agreement is found between the measured cumulative length and the range of cumulative lengths computed with formulae of Silvester and Rand. For VLB G5 a hydraulic jump is formed in the stilling basin, independent of the tailwater depth. As a consequence, the measurements of the end of the hydraulic jump pertain to a lower tailwater depth, in comparison to the conjugate water depth for the other geometries. 7 HIGHER TAILWATER DEPTHS The formulae and results discussed in sections 4, 5 and 6 are valid for a tailwater depth corresponding to the conjugate water depth. In this section, results belonging to higher tailwater depths will be considered.

Figure 15. Variation of near bottom velocity in function of tailwater depth – Geometry BGM G1, aerated, with $d_c/\Delta z = 0.645$ – for 3 distances $x/\Delta z$ downstream of the drop.

7.1 Formulae

Stilling basin design formulae in literature are based on a design discharge. The conjugate water depth is then computed based on this discharge value and the structure geometry. When following this design strategy, a stilling basin is designed with downstream (polder) water depths more or less equal to the conjugate water depth. For higher tailwater depths than the conjugate water depth, no suitable literature formulae were found by the authors. However, the combined inlet and outlet structures for the FCA-CRT's will have to deal with a broad combination of upstream (river) and downstream (polder) water levels. Consequently, the formulae discussed in sections 4, 5 and 6 are not well suited and no comparison with literature formulae will be made in this section.

7.2 Results

For each of the three types of stilling basin geometries (a straight stilling basin, a locally deepened stilling basin and a stilling basin using baffle blocks), as an example, only a selection of scale model results will be presented. This selection will be limited to the vari ation of the near bottom velocity and the variation of the flow pattern in function of the tail water depth.

7.2.1 Straight stilling basin

For BGM G1, the variation of the near bottom veloc ities at three distances downstream of the drop are presented in Figure 15 as a function of the tailwater depth.

Figure 15 shows a rapid transition between super critical and subcritical flow speeds in the range H TWL /Δz = 1.10 to 1.15. After this sudden drop, the near bottom velocity recovers somewhat. However, for tailwater depths exceeding 1.4, the near bottom velocity gradually decreases.

7.2.2 Locally deepened stilling basin

Figure 16 presents an example of the influence of the roof on the flow pattern for a geometry with a locally deepened stilling basin. Figure 16. Influence of roof on flow pattern (Geometry VLB G5 with $d_c/\Delta z = 0.373$). From Figure 16 follows that the falling nappe trajectory overtops the end sill for a tailwater depth $H_{TWL}/\Delta z = 1.14$. At this condition, the variation of the near bottom velocity in function of the tailwater depth shows an increasing velocity, as shown in Figure 17. These increased near bottom velocities are present in a rather narrow range of tailwater depths, from $H_{TWL}/\Delta z = 1.11$ to $H_{TWL}/\Delta z = 1.16$. Beyond a tailwater depth $H_{TWL}/\Delta z = 1.16$, the falling nappe makes contact with the roof of the combined inlet and outlet structure. The near bottom velocity then decreases again and resumes similar values as prior to the increase. 7.2.3 Baffle blocks Figure 18 presents an example of the variation of the flow pattern in presence of baffle blocks for 3 tailwater depths. From Figure 18 follows that an increase of the tailwater depth results in a decrease of the angle of the nappe (with the horizontal). Consequently, at a certain tailwater depth the location where the falling nappe touches the bottom is situated downstream of the location of the baffle blocks. The further increasing of the tailwater depth results into a contact of the falling nappe with the roof of the culvert and a redirection of the flow towards the bottom of the stilling basin, (Figure 18 bottom panel). This effect is also visible in the variation of the near bottom velocity in function of the tailwater depth, which is shown in Figure 19.

Figure 17. Variation of the near bottom velocity in function of tailwater depth (Geometry VLB G5, non-aerated, with $d_c/\Delta z = 0.373$) for $x/\Delta z = 3.3$ and $x/\Delta z = 5.3$ downstream of the drop.

Figure 18. Influence of baffle blocks on flow pattern (Geometry BGM G8 with $d_c/\Delta z = 0.645$).

The near bottom velocity first increases with an increasing tailwater depth and then decreases when the falling nappe makes contact with the roof.

8 CONCLUSIONS

In the framework of the design of Flood control Areas (FCA) with a Controlled Reduced Tide (CRT) for the

Updated Sigmaplan, Flanders Hydraulics Research

performed a scale model based review of different

desktop designs of combined inlet and outlet struc

tures. This paper compares scale model measurements

and the predictions of literature formulae for drop

structures and stilling basins. Three types of geome

tries are discussed: a straight stilling basin, a locally

deepened stilling basin and a stilling basin with baffle

Hager, W.H.; Bremen, R.; Kawagoshi, N. (1990). Classical

hydraulic jump: length of roller. J. Hydraul. Res. 28(5):

591–608. doi:10.1080/00221689009499048.

Peterka, A.J. (1984). Hydraulic design of stilling basins and

energy dissipators. Engineering monograph (Washing

ton), 25. U.S. Dept. of the Interior, Bureau of Reclamation:

Washington.

Rand, W. (1955). Flow geometry at straight drop spillways.

Proc. Am. Soc. Civ. Eng. 81: 1–13.

Silvester, R. (1964).Theory and Exepriment on the Hydraulic

jump. Proc. Am. Soc. Civ. Eng. J. Hydraul. Div. vol. 90,

n0 HY1 23–55. Vercruysse, J.; De Mulder, T.; Verelst, K.; Peeters, P. (2013). Stilling basin optimization for a combined inlet-outlet sluice in the framework of the Sigmaplan. Proc. of Int. Workshop on Hydraulic Design of Low-Head-Structures (IWLHS 2013), Karlsruhe, Germany: Bundesanstalt für Wasserbau (BAW), p. 55–66. Thompson, P.L.; Kilgore, R.T. (2006). Hydraulic design of energy dissipators for culverts and channels. Hydraulic Engineering Circular, 14. U.S. Department of Transportation. Federal Highway Administration: Arlington.

287 pp. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Supercritical flow around an emerged obstacle: Hydraulic jump or wall-jet-like bow-wave?

G. Vouaillat, N. Rivière, G. Launay & E. Mignot

Université de Lyon, INSA de Lyon, Laboratory of Fluid Mechanics and Acoustics, France

ABSTRACT: When an impervious, emerged, bluff obstacle is placed within a supercritical flow, it forces the flow to skirt it. The workaround can take two distinctive forms. A first one is the “detached hydraulic jump”, similar to the detached shock waves that form upstream of bluff bodies in supersonic flows. A second form of workaround is the so-called “wall-jet-like bow-wave”, which is in fact more commonly encountered in the field around bridge piers within supercritical rivers. In this case, the workaround is directed upwards instead of laterally. The present work, through laboratory experiments, details the two forms – jump and wall-jet – and focuses on the conditions of appearance of one form or another, in terms of dimensionless parameters.

1 INTRODUCTION

Emerging obstacles embedded in a supercritical flow are encountered in many field situations: emerging boulders in steep rivers, buildings within the inland flow caused by a tsunami, bridge piers in supercritical rivers in piedmont plains, vehicles in sloping streets during urban floods, avalanche protection devices, etc. Due to the properties of a supercritical flow, the workaround of the obstacle would require a detached hydraulic jump to form. However, a second type of workaround – without formation of any hydraulic jump – is also observed, the so-called “wall-jet-like bow-wave”.The objective of this work is to study these two ways of workaround used by the supercritical flow to skirt the obstacles and their condition of appear ance. The paper is organized as follows. After this introduction, section 2 reminds some knowledge about the properties of supercritical/supersonic flows, their implications when skirting obstacles and gives detail on the two forms of workaround. Section 3 explains the experimental methods. Section 4 provides the results, followed by conclusions.

2 SUPERCRITICAL FLOW AND OBSTACLES

2.1 Brief reminder about supersonic flows

Widely spread results concerning the deflection of a supersonic gas flow by an obstacle are worth reminding, as they present strong analogies with our open-channel supercritical flow problem.

When an impervious obstacle forces a compress ible flow to deviate, the corresponding workaround adopts different forms, depending on the flow regime and on the obstacle shape. When the flow is subsonic, its velocity is smaller than the speed of sound: disturbances created by the presence of the obstacle can travel upstream. They cause a curvature of the streamlines that ensures the lateral flow deflection. When the flow is supersonic, its velocity is higher than the speed of sound and disturbances created by the presence of the obstacle cannot go back up the flow. The deflection is thus sudden, through a shock wave that adopts two forms. When the deflection angle required to skirt the obstacle is small enough (case of slender bodies), an oblique shock wave performs the deflection. The streamlines remain straight lines but experience an abrupt change of direction through the shock. If the deflection angle is important (case of bluff bodies), a detached shock wave forms upstream from the obstacle. Within the zone delimited between the obstacle and this shock, the flow becomes subsonic and anticipates the presence of the obstacle: the curvature of the streamline is possible and performs the lateral deflection of the flow. 2.2 Supercritical open-channel flows and detached hydraulic jumps Similar phenomena characterize the deflection of an open channel flow by an emerged obstacle. If the flow is subcritical, it deviates progressively through a streamlines curvature, but which is additionally combined with backwater effects. If the flow is supercritical, such gradual phenomena are not possible as the flow velocity is higher than the celerity of the gravity waves. If the deflection angle is small enough (case of an emerged, sharp obstacle), it is carried out by an oblique hydraulic jump. High angle deflections (case of bluff emerged obstacles) require the formation of a detached hydraulic jump upstream from the obstacle.

Figure 1. Detached hydraulic jump: lateral section and

photograph of an example.

Such detached jumps were indeed observed in previ

ous studies (Defina & Susin, 2006; Mignot & Rivière,

2010; Mignot et al., 2016).The photograph of Figure 1

provides an example. The jump toe forms a hyperbola

located upstream from the obstacle, with a detachment

length λ jump of several times the water depth. As the

supercritical flow crosses the jump, it experiences an

abrupt water depth increase and becomes locally sub critical. The backwater curve is pronounced, due to the presence of a stagnation point (also observed in 1D flows and called “impact-hydraulic jumps” by Hager, 1994) which causes the formation of a small bow wave, as observed around bridge piers in subcritical rivers (Richardson & Panchang, 1998).

2.3 Wall-jet-like bow-wave

However, in the field, the flow observed around blunt obstacles such as bridge piers or boulders in supercrit ical rivers can take a quite different form, which con tradicts apparently the preceding section on detached jumps. This form is named herein “wall-jet-like bow wave”, i.e. a bow-wave formed by an upward wall-jet on the obstacle upstream face. Photograph of Figure 2 depicts such a flow which is significantly different from a detached jump. The flow deviates abruptly upward, very close to the obstacle, at a distance of about one water depth h upstream from the obstacle, which is also the scale of the radius of curvature at the jet basis. A vertical wall-jet forms on the upstream face of the obstacle, similar to impinging liquid jets in

Figure 2. Wall jet like bow wave : lateral section and photograph of an example.

air (e.g. Wilson et al., 2012). On the obstacle face, the upward jet separates towards both sides of the stagnation point and forms two lateral jets outing in a top-side diagonal direction. The jet, in its upper part, in front of the obstacle, has the form of a breaking bow-wave with a reverse spillage, which causes periodic oscillations of the jet. Indeed, as water from this spillage falls into the supercritical flow, upstream from the obstacle, it suddenly reduces the kinetic energy of the flow, and so the water elevation at the stagnation point. This suppresses the spillage: the upstream flow recovers its initial kinetic energy and this is the beginning of a new cycle. With these wall-jet-like bow-waves, the deflection of the jet is directed in the upward direction. In other words, the discharge blocked by the obstacle is deviated outside from the flow, where it is reintroduced downstream from the obstacle, disregarding of the possible reverse spillage.This is completely different from the hydraulic jump case, where the blocked discharge is deviated laterally and always remains within the main flow. As the two flow forms exist, it is worth seeing the condition of appearance of a wall-jet-like bowwave instead of a detached hydraulic jump. This is undergone herein using experiments. 3 EXPERIMENTS 3.1 Dimensional analysis Design of experiments benefits from the use of dimensional analysis. Considering the present problem of an emerged obstacle in a supercritical regime,

Figure 3. Top-view of the water table (facility 1).

seven dimensional variables characterize the flow.

Indeed, supercritical uniform flows in rectangular sec

tion open-channels are characterized by: the uniform

upstream water depth h, the upstream mean velocity

U , the water properties (density ρ, dynamic viscos

ity μ and surface tension with air σ) and the gravity

acceleration g. Only emerged obstacles are considered

herein, characterized for parallelepipedal obstacles by

the front-surface width D or for round obstacles by the

diameter D. As the obstacles are always emerged, their

height is not considered. More, as the flow is supercrit

ical, the obstacle length (in the streamwise direction)

is not considered as it only modifies the wake. Finally,

the channel width B is not considered: it is always large enough compared to the obstacle width D so that disturbances induced by the presence of the obstacle reach the walls only downstream from the obstacle (see Defina and Susin, 2006), again with no influence on the flow around the obstacle in a supercritical flow. Vaschy-Buckingham's #-theorem, using h as length scale, h/U as time scale andph 3 as mass scale, provides four dimensionless parameters that rule the flow: where h/D is the depth to obstacle width ratio, "Fr" the upstream Froude number, and "Re" the upstream Reynolds number based on the upstream water depth (justified notably when considering h B). "We" is the Weber number that accounts for capillary effects due both to surface tension and local water/air inter face curvature. D and h are length scales of the radius of curvature of the wall jet in the horizontal plane and in the vertical symmetry plane, respectively. The minimum of these two values, min (h, D) is logically retained to define We and therefore to account for the strongest capillary effects which correspond to the smallest radius or curvature and thus to the smallest

We values (estimated here using σ = 0.076 N/m). Figure 4. Top-view of the open-channel (facility 2). Table 1. Characteristics of the two facilities. Facility 1 Facility 2 Facility (water table) (channel) Width B (m) 0.75 0.25 Length L (m) 1.2 9.24 Q v (L/s) 0.25–3.2 0.51–21.89 �Q v ±0.05 L/s ±1% h (mm) 1.28–12.33 10–50 �h (mm) ±0.25 ±0.5 D (mm) (square obst.) 13–100 10–50 (round obst.) 25–76 h/D

0.012-2.57 0.3-4 Fr 1-6.35 1-2.5 Re 1300-16000 10000-208000 We 1-50 5-800 3.2 Experimental facilities Two different facilities are used in order to obtain a significant range of the dimensionless parameters above. Facility one is a water table which allows small D/h ratios. Facility two is a more classical open-channel characterized by higher D/h ratios. Table 1 sums-up the range of the parameters characterizing the experiments in the two facilities : channel length L, channel width B, range of obstacle width D, discharge Q v and associated uncertainties, upstream flow depth h and associated uncertainties. Follow the corresponding values of the four dimensionless parameters.The appearance of one form rather than the other is expected to depend on these four parameters: this is addressed in the next section. 4 EXPERIMENTAL RESULTS 4.1 Occurrence of one form or another The condition of appearance of one form rather than the other is correlated to the dimensionless flow parameters by observing the form obtained under a large number of experimental conditions, including both supercritical and subcritical flows. Stating if the flow is of one form or the other is somehow subjective. It was considered that when oscillations of the water depth on the obstacle front face are observable, they indicate a spillage and the presence of a wall-jet-like bow-wave. The results are plotted on Figure 5, in a (Fr, h/D) plane. The open symbols correspond to a wall-jetlike bow-wave. The closed symbols correspond to a

Figure 5. Flow forms observed depending on h/D and Fr:

detached hydraulic jump (closed symbols) and wall-jet-like

bow-wave (open symbols). Experiments using facility 1

sketched by triangles and using facility 2 by circles.

detached hydraulic jump. Two populations are clearly

distinguishable: wall-jet in the upper and right part of

the graph and hydraulic jump in the lower and left part

of the graph. Experiments were performed in facility 1

(triangles) corresponding to moderate values of both

Re and We, while others were performed in facility

2 (circles) corresponding to higher Re and We val

ues. Considering the transition somehow subjective, the overlap of the two datasets (triangles and circles) indicate a small influence of both Re and We. The appearance of a detached jump or of a wall-jet-like bow-wave mainly depends on the two parameters Fr and h/D For a given Froude number, the wall-jet-like bow-wave occurs for high h/D ratios and the detached jump for smaller ones. For a given h/D, the wall-jet-like bow-wave occurs for high Froude numbers. As a brief, the increase of both h/D and Fr favours the appear ance of the wall-jet-like bow-wave. The transition corresponds satisfactorily with the curve:

4.2 Influence of the obstacle shape

The dependency of the appearance of one form or the other on Fr and h/D is established for paral lelepipedal (rectangular section) obstacles. Additional experiments are undergone to examine the influence of more streamlined obstacles, namely with a round section. Corresponding results are plotted on Fig ure 6 (squares), along with previous results with a square section (circles). The transition corresponds satisfactorily with the curve:

in dotted line on Figure 6. For small Froude numbers,

Defina A. and Susin F. M. 2006. Multiple states in open channel flow, in vorticity and turbulence effects in fluid structures interactions. Advances in Fluid Mechanics, edited by M. Brocchini and F. Trivellato, WIT Press, Southampton, UK, 2006 pp. 105–130.

FaugT., Childs P.,Wyburn E., Einav I. (2015). Standing jumps in shallow granular flows down smooth inclines, Physics of Fluids, 27, 073304.

Graf, W. H., and Altinakar, M. S. (2000). Hydraulique Fluviale, Traité de Génie Civil, vol. 16. Presses Polytech niques et Universitaires Romandes.

Hager, W. H. (1994). Impact hydraulic jump. Journal of Hydraulic Engineering, 120(5), 633–637.

Mignot, E., Rivière, N., Perkins, R. and Paquier, A. (2008). Flow patterns in a four-branch junction with supercritical flow. Journal of Hydraulic Engineering, 134(6), 701–713.

Mignot, E. and Riviere, N. (2010). Bow-wave-like hydraulic

jump and horseshoe vortex around an obstacle in a supercritical open channel flow. Physics of Fluids, 22(11), 117105. Mignot E., MoyneT., Doppler D., Riviere (2016). Clear-water scouring process in a flow in supercritical regime. Journal of Hydraulic Engineering. 142(4), 04015063. Richardson, J. and Panchang, V. (1998). Three-Dimensional Simulation of Scour-Inducing Flow at Bridge Piers. J. Hydraul. Eng., 124(5), 530–540. Rivière N., Laïly G., Mignot E., Doppler D. (2012). Supercritical flow around and beneath a fixed obstacle. 2nd IAHR Europe Congress, June 27-29 2012, Munich, Germany. Wang, T., Davidson, J. F., & Wilson, D. I. (2013). Effect of surfactant on flow patterns and draining films created by a static horizontal liquid jet impinging on a vertical surface at low flow rates. Chemical Engineering Science, 88, 79–94. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Monsin movable dam in Belgium: A case study

C. Swartenbroekx, C. Savary & D. Bousmar

Hydraulic Research Laboratory, Service Public de Wallonie, Châtelet, Belgium

ABSTRACT: Monsin movable dam is located on river Meuse, in Belgium, and allows for navigation between Liège, Maastricht and Anvers. The Walloon administration is considering its renovation since the present dam was built in 1930. Each of the 6 openings is closed by a vertical lift valve on top of which is sat a flap gate. The ratio between the heights of these elements will be slightly modified to ease the regulation. Several hydraulic behaviours had then to be checked. A 1:15 scale physical model of a slice of this dam was built in a 1 m wide flume, according to Froude similarities. It was used to determine (i) the efficiency of the aeration of the overflowing nappe; (ii) the relationship between the discharge and the valve position; (iii) the hydrodynamic pressure distribution on the valves; and (iv) the energy dissipation on the downstream concrete apron.

1 INTRODUCTION

Monsin movable dam is located on river Meuse, down stream of the city of Liège, in Belgium. It allows the water level in Albert canal to be maintained. It is thus a key dam for navigation between Liège and the harbor of Antwerp, and between Belgium and The Netherlands. It also regulates the water flow during high stage periods in such a way that it protects the city of Liège from floods. It is the main structure con trolling the flow in the area of Liège (Dewals et al.

2015). Moreover, there is a hydropower plant next to the dam. The Walloon administration is consider ing a renovation since the present dam, built in 1930, is aging and adequate safety of the structure can no longer be guaranteed.

It is made of 6 27 m-wide sluices. Each opening is closed by a vertical lift valve on top of which is sat a flap gate. Depending on the discharge (up to 2500 m^3/s), the water can run as an overflow, an underflow or a mixed flow. During usual discharges (Fig. 1a), the flap gate is tilted appropriately to control the overflow and the upstream water level. For higher discharges, the flap gate is at its lower position and the valve has to be lifted to allow for mixed flow (Fig. 1b) and, eventually, underflow (Fig. 1c). The present flap gate is 1.96 m-high and the lift valve is 3.39 m-high. These heights will be slightly modified (becoming 2 m and 3.35 m, respectively), while maintaining an over all structure of 5.35 m-high.These new ratio will allow for more overflow, which is easier to regulate than underflow. Due to this modification, several hydraulic behaviors had to be checked.

A 1:15 scale physical model of a slice of this dam was built in a 1 m-wide flume, according to Froude similarities. It was used to determine (i) the effi ciency of the aeration of the overflowing nappe; (ii) the Figure 1. Lateral view of vertical lift valve and flap gate at (a) overflow, (b) mixed flow, and (c) underflow (physical model). relationship between the discharge and the valve position; (iii) the hydrodynamic pressure distribution on the valves; and (iv) the energy dissipation on the downstream concrete apron. 2 EXPERIMENTAL SET-UP The physical process was analyzed thanks to a scale model, built in the Hydraulic Research Laboratory of the Walloon administration. By notation, the index p is related to prototype or field data while the index m corresponds to physical model values.The x-, y-, and z-axis are the streamwise,

the transverse and the upward vertical axis, respec

tively. The x-axis starts at the valve foot and the z-axis

at the downstream floor level.

2.1 Model scale

Due to space and discharge constraints, the chosen

scale was 1:15 according to Froude similarities (Kobus

1980). It was checked that the flow is kept turbu

lent at this scale, with a Reynolds number Re highly

superior to 2000. Moreover, in the worst case of an

overflow of h m = 4 cm, theWeber number isWe = 181,

which means that the surface tension gap between the

prototype and the model does not impact the flow.

2.2 Flume characteristics

The flume is 40 m long, 1 m wide and 1.5 m high. The

walls are made of glass. The water supply is realized

by the laboratory’s water recirculation pipe system.

Figure 2. Apron at prototype scale (side view of a central

section).

Figure 3. Apron and energy dissipation structures in the

field (upstream view). 14 structures lie on the apron in the

central part, 12 structures lie in the lateral parts.

Figure 4. Side view of the physical model during a mixed flow. From left to right: the upstream water level, the valve, the

flow over the downstream apron and the pit with mobile bed. An upstream tank and an inlet section allow for a homogeneous inlet flow. The incoming discharge can vary between 20 l/s and 400 l/s. The tailwater level is regulated via an outlet flap gate at the end of the channel. 2.3 Vertical lift valve and flap gate The projected heights of the new dam will be $h_{g,p} = 2$ m for the flap gate and $h_{v,p} = 3.35$ m for the lift valve. At the model scale, it means $h_{g,m} = 0.133$ m and $h_{v,m} = 0.223$ m, respectively. In the lab, the valve is lifted and the flap gate is tilted via a system of lines and pulleys connected to a servomotor. Two vertical runners guide the vertical lift and are embedded in the channel walls. The remaining available width for the valve is $w_m = 0.926$ m, which stands for a slice of the dam of width $w_p = 13.89$ m in the field, i.e. almost half of an opening. The inlet total discharge in the model accounts for this reduced width to maintain an equivalent unitary discharge between model and prototype. 2.4 Concrete floor The valve is located at the level $z_{b,p} = 54.9$ m above the sea level in the field, while the downstream apron level is $z_{b,p} = 54.3$ m. There is thus a step of $h_{b,p} = 60$ cm. The total apron length is $l_p = 53.66$ m. Fourteen structures, with a squared section of 0.3 m × 0.3 m, separated by 2.5 m, lie on the apron to dissipate the energy (Fig. 2). They lie on the whole cross-section of the dam, except the structures number 4 and 5 that are missing in the continuity of the four lateral dam openings (Fig. 3). During a field inspection in 2013, the apron quality was considered satisfactory. It does not need to be renovated from a structural point of view. The modeled apron is shown in Figure 4. The gate is located on a step of �$h_{b,m} = 4$ cm and the dissipation structure section is 0.02 m × 0.02 m. A pit of 0.4 m deep is located downstream of the concrete floor at coordinate $x_m = 3.58$ m, to install a mobile bed. 2.5 Sediment features In the field, according to geotechnical survey realized in 2002 and 2008 downstream of Monsin, the bed of river Meuse is made of schist bedrock between the level $z_{b,p} = 50$ m and $z_{b,p} = 53$ m. Above this level, the natural bottom is mainly made of sand and silt.

The bathymetry was measured with sonar equipment in March 2013 (Fig. 5). The observed erosion expands over 140 m. The deepest scour (deep blue area) reaches the level $z_{b,p} = 46$ m locally (8 m-deep in comparison with the concrete floor level) and occurs about $x_p = 50$ m downstream of the floor. A deposition crest (red area) is then observed, reaching the apron level. The natural bottom should be considered at the mean level $z_{b,p} = 50$ m (cyan area).

It is not easy to simulate the bedrock at this natural level in the physical model. According to Shields diagram, a critical grain (stable at the level $z_{b,p} = 50$ m and unstable at the level $z_{b,p} = 52$ m) for the highest discharges is represented in the laboratory by gravel 6/14 with a median diameter $d_{50,m} = 10$ mm, as a first approach.

No data are available about the field solid discharge, but it is assumed weak on the apron. There is thus no upstream sediment feeding in the model.

2.6 Flow

In the river Meuse, the water flow discharge may reach $Q_p = 2500$ m^3/s exceptionally. However, a discharge $Q_p = 800$ m^3/s occurs usually less than 15 days a year. Figure 6 gives the frequency of peak discharges measured during the last decade in Visé, some kilometers downstream of Monsin dam. The recurrence time per

year can be computed by multiplying the frequency by

365 days.

Figure 5. Bathymetry in Monsin, measured in 2013.

Figure 6. Frequency of flood discharges in the river Meuse

(measured in Visé). The normal waterline for navigation is between level 60 m and 60.15 m above the sea level in the Albert canal. In the downstream Meuse, the waterline depends on the water level at the downstream Lixhe dam. The relationship between the discharge and the water level in Monsin is known thanks to field measurements (Fig. 7). The downstream water level ranges between $z_{w,p}$ = 54.4 m and $z_{w,p}$ = 58.6 m. 2.7 Measurements equipment The discharge was measured by means of two electromagnetic flowmeter installed in the supply lines, with an accuracy of 0.2%. The water level was measured in six fixed locations with ultrasonic gauges. The water depth h_m ranges between 2 and 50 cm. The sensor accuracy is claimed to equal 0.3 mm. In addition, an ultrasonic gauge was located on a trolley. The position of this trolley is recorded via a laser distance sensor. Twenty pressure plugs were located on the valve, as shown in Figure 8.There was also a pressure plug in the floor, located 6 m upstream of the valve, considered as a hydrostatic pressure plug taken as a reference. These pressure plugs could be connected, via small flexible conduits and taps, to a differential pressure gauge with Figure 7. Upstream and downstream measured water level in Monsin. Figure 8. Position of the pressure plugs on the valve.

a range of +/-50 mbar and an accuracy of 0.05 mbar.

Before each test, water was pumped in the conduits to

be sure that no air is remaining. A Preston gauge was

also used to measure the static pressure component,

at the same level as the pressure plugs and connected

in a same way to the differential pressure gauge, for

comparison.

The bed profile was measured with acoustic probes

fixed on the trolley, during and at the end of the ero

sion process. The probe was slowly displaced along

the channel to cover the whole model pit area.

The valve and gate opening was controlled by

servo drives. The data acquisition and the opening

instructions were handled by means of the software

HydroCap 3, a home-made environment developed

with Labview (Bousmar 2008).

3 AERATION OF THE OVERFLOWING NAPPE

A proposal was made to aerate the overflowing

nappe permanently and to avoid flap gate oscilla

tions (Swartenbroekx & Libert 2015). A lateral nappe

spoiler and six breakers were placed on the upstream

face of the gate as shown in Figure 9.They were

Figure 9. Physical model of nappe spoiler and breakers

(downstream view).

Figure 10. Overflow on nappe spoiler and breakers (down

stream view). tested qualitatively in the lab.Appropriate aeration was obtained and no oscillation was observed (Figs 10- 11). Then, at field scale, the lateral spoiler should be about 1 m-high while the breakers should be less than 1 m-long and located every 2 m. 4 DISCHARGE AND VALVE POSITION An inlet flow increasing from Q m = 20 to 290 l/s, by steps of 5 l/s, was imposed in the model (Swartenbroekx & Libert 2015). The corresponding downstream water level was imposed via the outlet gate, since it impacts the mixed flow and underflow around the lift valve. The flap gate and then the lifted valve openings were adjusted at each step in order to respect the water level required at the upstream reach for navigation. Because the upstream reference may slightly vary, three upstream levels were tested, corresponding to: (a) z_p = 60.075 m, (b) z_p = 60.150 m, and (c) z_p = 60.225 m. The results were converted to prototype scale in Figure 12, assuming that the six valves are opened simultaneously, which is usually not the case. The valve opening corresponds to: (i) the level of the Figure 11. Mixed flow on nappe spoiler and breakers (downstream view). Figure 12. Valve opening per discharge (prototype scale).

downstream extremity of the flap gate in case of an overflow; and (ii) the level of the lower point of the lift valve in case of a mixed or underflow.

The limit between overflow and mixed flow occurs for Q_p = 820 m^3/s, i.e. less than 4% of time, while an underflow starts for Q_p = 2135 m^3/s, i.e 0.02% of time (Fig. 6). These limits are bigger than the present val ues, as expected.These estimates have to be considered cautiously. The dam management is indeed more com plex, accounting for the hydropower plant discharge and allowing for a distinct treatment between the six openings.

5 HYDRODYNAMIC PRESSURE PROFILE

A first test was made with still water to check the effi ciency of the pressure measuring equipment (Savary & Libert 2015). The pressure plugs 1, 4, 13 and 17 were detected to be damaged and were thus neglected.

Several configurations were considered; three are here presented: (i) an overflow with the flap gate in a position such that the tangent to the extremity of the flap gate is horizontal, corresponding to Q_p = 385 m^3/s

(Fig. 13a); (ii) an overflow with the flap gate in its lower

Figure 13. Position of the pressure plugs on the valve.
position corresponding to $Q_p = 820\ m^3/s$ (Fig. 8); and (iii) a mixed flow corresponding to $Q_p = 1435\ m^3/s$ (Fig. 13b). Figure 14 exposes the hydrodynamic pressure profile measured at each pressure point on the valve model, transposed at prototype scale. No significant difference is observed between the measurements realized with the pressure plugs (red dots) and the Preston gauge (green dots), giving satisfactory confidence in the measuring system. The measurements were then compared to the hydrostatic pressure that would be obtained if the water level was horizontal and equivalent to the level measured by the upstream ultrasonic gauge (black lines). In the case of overflow, a higher discharge and a lower flap gate implies a bigger gap between the hydrostatic profile and the hydrodynamic profiles: the hydrodynamic pressure is decreasing on the flap gate and the top of the lift valve (Fig.14b in comparison with Fig.14a). In case (ii) where the flap gate is at Figure 14. Measured pressures along the valve at prototype scale: (a) overflow, (b) overflow and (c) mixed flow.

Table 1. Resulting forces and momentum on the flap gate

(for a slice of 1 m-wide gate at prototype scale).

	Castro abacus		Measurements	
Test	F_z (kN)	M_o (kNm)	F_z (kN)	M_o (kNm)
(i) overflow	24.9	28.5	21.9	28.9
(ii) overflow	9.4	7.9	11.9	14.76
(iii) mixed flow	7.3	6.9	8.6	10.9

its lower position, there is almost no pressure on the extreme point 12 and water can flow freely (Fig. 14b).

In case of mixed or underflow, a gap between the hydrostatic profile and the hydrodynamic profiles is appearing at the valve toe (Fig. 14c). Stronger the flow is beneath the valve, stronger the gap is.The water level is then locally raised near the valve, due to recirculation processes.

The pressure profiles were integrated to compute the resulting vertical force F_z acting on the flap gate as well as the momentum M_o acting on the tilting axis of the flap gate. The results (given for a slice of 1 m-wide gate at prototype scale in Table 1) were then compared to Castro abacus as proposed by Naudascher (1991). They are of the same order of magnitude, with theoretical values inferior to the experimental ones.

6 DOWNSTREAM ENERGY DISSIPATION

Both the waterline and the mobile bed erosion were measured downstream of the valve, in order to check how the fall energy is dissipated.

6.1 Waterline and hydraulic jump

Just downstream of the valve, the flow is supercritical. Its energy is dissipated further by a hydraulic jump. This energy dissipation has to occur above the apron to avoid scouring in the downstream Meuse.

The commonly expected purpose of energy dissipation structures on an apron is to shorten the longitudinal range within which the hydraulic jump will take place (Chow 1973) and to ensure the apron is long enough.

In Monsin case, the apron level is uncommonly high compared to downstream water level. This is due to historical reasons, with the removal of a downstream dam and a modification of the reference water level. The structure impact on the waterline was checked by Vanormelingen (2013), Durvaux (2014), and Bodart (2015).

Figure 15a displays an overflow when $Q_p = 681\ m^3/s$ while Figure 15b shows a mixed flow when $Q_p = 1096\ m^3/s$ and Figure 15c displays an under flow when $Q_p = 2098\ m^3/s$.Three configurations were compared: (i) an apron with 14 structures (blue); (ii) an apron with 12 structures, i.e. dissipaters 4 and 5 skipped (green); and (iii) no structures (red). The computed critical depth is drawn (magenta), to detect where is located the hydraulic jump. The waterline could not be measured just downstream of the valve due to eddies created by the plunging flow. With the 14 dissipaters, the hydraulic jump is located at $x_p = 15.75$ m for $Q_p = 681\ m^3/s$ (Fig. 15a), at $x_p = 6.45$ m for $Q_p = 1096\ m^3/s$ (Fig. 15b), and at $x_p = 29.25$ m for $Q_p = 2098\ m^3/s$ (Fig. 15c). The maximum depth is $z_m = 185$ mm, $z_m = 200$ mm, and $z_m = 210$ mm at model scale ($z_p = 57.08$ m, $z_p = 57.3$ m, and $z_p = 57.45$ m at prototype scale), respectively. The fluctuating waterline downstream of the apron is occurring on a longer path with a higher discharge. When there are 12 dissipaters, the waterline is not very distinct: it is only modified locally with a smaller water depth upstream of the usual position of dissipater 4 and thus a higher velocity. Without dissipaters, there is no hydraulic jump and the flow is going on being supercritical downstream of the apron. For all tested flows, the hydraulic jump stayed located above the apron if there are at least 12 dissipaters. In conclusion, the present apron and dissipation structures are efficient enough and necessary to dissipate the energy that will be created by the new valve and flap gate.

6.2 Bed erosion

The time evolution of the bed profile was measured downstream of the apron for the same cases. The results are presented at time t_m = 70 minutes after the beginning of the flow, and for a profile at the channel centre, at model scale (Fig. 16). They should be considered cautiously due to the hypotheses made to choose the sediments in the model and because the channel walls have a significant influence on the crosssections. The physical model allows for a comparison between configurations tested in similar conditions; but the bed levels should not be extrapolated to field scale. During an overflow, for small discharges, no sediment transport was detected. When Q_p = 681 m^3/s (Fig. 16a), there was a scour hole occurring at the transition between the apron and the mobile bed; the gravels were deposed a little bit farther. The bed was then quasi horizontal.The same behavior was observed with and without the dissipaters. The solid transport was stronger for the mixed flow (Fig. 16b) and for the underflow (Fig. 16c). In the case of underflow (Fig. 16c), especially without dissipaters, the observed deposition crest in the central part of the channel was mainly due to solid transport in the crosssection: there were helical vortices along the side walls eroding the bed. During rapid flows, the worst cases appeared in the absence of dissipation structures. This confirms the necessity to maintain the dissipaters on the apron.

Figure 15. Waterlines at model scale for (a) the overflow when Q_p = 681 m^3/s, (b) the mixed flow when Q_p = 1096 m^3/s, and

(c) the underflow when Q_p = 2098 m^3/s.

7 CONCLUSIONS

The Monsin movable dam is essential for navigation

to be allowed in Albert canal and to control the water

level of river Meuse in Liège area. It is planned to

renovate the six valves, modifying their shape and

the ratio between the flap gate height (2 m instead of

1.96 m) and the lift valve height (3.35 m instead of

3.39 m). This ratio modification will impact several

hydraulic behaviors, which were analyzed thanks to a

scale model.

The overflowing nappe must be aerated. The aera

tion can be guaranteed with a lateral spoiler and several

breakers. The relationship between the flow discharge and the valve openings will be affected by the new valve geometry, allowing for more overflow as expected. The limit between overflow and mixed flow occurs for about $Q_p = 820$ m^3/s while an underflow starts for about $Q_p = 2135$ m^3/s. These limits correspond to less than 4% of time and about 0.02 % of time, respectively. The hydrodynamic pressures exerted on the new valve were determined for structural design purposes. During an overflow, the hydrodynamic pressure is decreasing on the flap gate and the top of the lift valve in comparison with the hydrostatic profile. During an underflow, a gap between the hydrostatic profile and the hydrodynamic profiles is appearing at the

Figure 16. Bed level at model scale for (a) the over

flow when $Q_p = 681$ m^3/s, (b) the mixed flow when $Q_p =$

1096 m^3/s, and (c) the underflow when $Q_p = 2098$ m^3/s.

valve toe. This has to be accounted for structural

analysis.

An effect of the valve geometry change on the

energy to dissipate downstream of the valve is also

expected. However, the present apron and dissipation

structures are efficient enough to dissipate this energy,

Numerical modelling of contracted sharp crested weirs

A. Duru

İ ller Bank, Ankara, Turkey

A.B. Altan-Sakarya

Middle East Technical University, Ankara, Turkey

M.A. Kokpinar

State Hydraulic Works, Artvin, Turkey

ABSTRACT: Sharp crested weirs are flow measurement structures which are frequently used for discharge

measurements in channels and laboratories. In this study, numerical modelling technique was used in the solution

of flow measurement problems, namely contracted sharp crested weirs. The numerical simulation of flow cases

having similar conditions with the experiments from the previous studies is performed. The basic flow equations

are solved using the finite-volume method of the commercially available software Flow-3D. The volume of fluid

method is used to compute the free surface of the flow that interacts with the discharge measurement structures.

In the numerical analysis, RNG turbulence closure model is employed as turbulence model. The computed

results for the volumetric flow rate of the flow are compared with the experimental data. The comparisons of the

experimental and numerical results studied show that the computational volumetric flow rate values are found

to agree reasonably well with the experimental data.

1 INTRODUCTION

Control structures are one of the most important

devices in hydraulic engineering. The relationship

between the discharge and the water level is fixed by

channel control sections in open channels. Weirs are

commonly used hydraulic structures as control devices

to measure discharges. Sharp crested weirs are verti

cal obstructions placed normal to the flow direction.

Water passes through a contracted section over the weir plate. Sharp crested weirs have some advantages that the flow is substantially free from viscous effects and resultant energy dissipation (Henderson 1967; Aydın et al. 2011).

Analysis of flow over the weir is an important engineering problem. Therefore, recent developments in computer science and numerical techniques have advanced the use of Computational Fluid Dynamics (CFD) as a powerful tool for this purpose (Haun et al. 2011). Flow-3D is used as the numerical CFD simula tion software which applies the finite volume method to solve Reynolds Averaged Navier-Stokes (RANS) Equations.To represent the sharp interface between air and water, for free surface flows, the Volume of Fluid (VOF) method of Hirt & Nichols (1981) is used (Harg reaves, et al. 2007).This is a two phase approach where both water and air are modelled in the grid. The bor der between the geometry and the water is defined by

the Fractional Area Volume Obstacle Representation (FAVOR) Method (Haun et al. 2011). Also, the continuity and momentum equations of the fluid fraction are formulated using FAVOR function (Kermani & Barani 2014). In the scope of this study, the contracted sharp crested weir is studied numerically. The experimental results of the rectangular sharp crested weir were compared to the numerical results. The discharges that are computed numerically were compared to experimental ones. 2 LITERATURE REVIEW Discharge measurement is an important issue in hydraulic engineering. Weir is one of the flow measuring devices for open channels. They are defined as structures where the streamlines parallel to each other over the weir and the crest of the weir is horizontal. Weirs are used as discharge measurement structures, for example in hydropower structures, irrigation channels and laboratory flumes (Haun et al. 2011). Contracted sharp crested weirs are also vertical obstructions placed normal to the flow direction in which water passes through a contracted section over the weir plate (Aydın et al. 2011). Recent developments in computer science and numerical techniques have advanced the use of Computational Fluid Dynamics (CFD) as a powerful tool.

CFD is a type of numerical method used to solve problems involving fluid flow. Since CFD can provide a faster and more economical solution than physical models, engineers are interested in verifying the capa bility of CFD software. Some recent works show the capability of the CFD method in the numerical mod eling of flow over weirs (Kermani & Barani 2014). Ramamurthy et al. (2009) developed two-dimensional two-equation k-ε turbulence model together with the Volume of Fluid method to predict the characteristics of sharp crested weir in rectangular open channels. It had been stated that the predictions of the numer ical model agree well with the existing experimental and theoretical results. Arvanaghi and Oskuei (2011) had also studied sharp-crested weir discharge coef ficient by making use of CFD commercial software, Fluent and they reported a good conformity between the numerical and experimental results. Gharahjeh et al. (2015) obtained the numerical solution for the flow over the full width suppressed weir with vari able channel widths and water depths. They stated that computational results confirm the experimental findings. A 3D numerical model, Flow-3D had been used by Rady (2011) to estimate the discharge coef ficient for rectangular sharp crested full width weirs and stated that a good agreement with the literature was obtained. Other than rectangular sharp-crested weirs, Aydin and Emiroǧlu (2013) usedANSYS-Fluent mod els together with the laboratory models of labyrinth side weirs for determining the discharge capacity of the weir on straight channels. Samadi and Arvanaghi (2014) made the three dimensional simulation of flow on the contracted compound arched rectangular sharp crested weirs with Fluent software using RNG k-ε turbulence model and comparing with the experimen tal data it had been concluded that the simulation is achieved with high accuracy. In this study, also Flow 3D is used as CFD software tool, which is capable of simulating the dynamic and steady state behaviors of the fluid (Afshar & Hoseini 2013).

2.1 Contracted sharp crested weir

The weir is called sharp crested such that the head discharge relationship will not be affected from the crest length of the weir. For this condition to occur, the flow depth above the weir crest should be greater than about 15 times the crest length. The crest length

is normally less or equal to 2 mm. Rectangular sharp

crested weirs can be classified as partially contracted

and full width weirs (Bos 1989). In this study, a par

tially contracted weir is analyzed by using numerical

simulation. Since the weir is contracted from the sides

b/B ratio is reduced (Aydın et al. 2011).

For a rectangular weir, Equation (1) can be used as;

where Q = discharge (m^3/s), b = weir width (m),

h = head over the weir (m), g = gravitational acceler

ation (m/s^2), C_d = discharge coefficient (to represent
Figure 1. A typical view of a contracted sharp crested weir (Aydın et al. 2011). Figure 2. Side view of the contracted sharp crested weir (Aydın et al. 2011). all effects that are not taken into consideration such as viscous effect or streamline curvature). A typical view of a contracted sharp crested weir and its side view are given in Figures 1 and 2, respectively. In Figure 1, Bis the channel width (m), bis the weir opening (m), Pis the weir height (m), and h is the water level above the weir crest (m). Experimental data for the contracted weir obtained from the studies of S¸is¸man (2009) and Aydın et al. (2011) were used in the verification of numerical simulations. For the present study, effects of the different parameters (mesh size, turbulence model, upstream channel length) and formulation validation for the outside of the experimental range for this experiment were estimated with the help of the numerical simulations. For the discharge coefficient, C_d, calculations; Rehbock (1929); Kindsvater & Carter (1957); Swamee (1988); Ramamurthy et al. (1987); Bagheri and Heidarpour (2010a, 2010b) suggested different formulations in literature. The results taken from the numerical simulations were compared with the results of the formulation suggested by Aydın et al. (2011) since their experimental data is used. They suggested the following equations for the calculation of discharge as,

where A is cross sectional area of the flow and V_{wc}

is the average velocity at the weir section which is calculated by Equation (3).

where c_1 , c_2 and c_3 are the coefficients found from regression analysis given as,

3 NUMERICAL MODELLING

3.1 Formulation and numerical simulation

Fluid motion is described with non-linear, transient, second-order differential equations. The fluid equa tions of motion must be employed to solve these equations. The science of developing these methods is called computational fluid dynamics. Computational Fluid Dynamics (CFD) modelling employs specially developed numerical techniques to solve the equa tions of motion for fluids to obtain transient, three dimensional solutions to multi-scale, multi-physics flow problems. A numerical solution of these equa tions involves approximating the various terms with algebraic expressions.The resulting equations are then solved to yield an approximate solution to the original problem.The process is called simulation.The average values of flow parameters such as pressure and velocity are determined for each cell, with the help of gridding system. The representation of the boundary condition is also an important step for accurate simulation.

For three dimensional (3D) free surface flow sim ulations, the numerical solutions of RANS (Reynold

Average NavierStokes) equations are solved by using Flow-3D software based on FVM (Finite Volume Method) in a Cartesian, staggered grid (Hirt & Nichols 1981). Flow-3D is commercially available CFD soft ware, capable of solving a wide range of fluid flow problems.

The governing continuity and RANS equations for Newtonian, incompressible fluid flow are (Ozmen Cagatay & Kocaman 2010):

where u i represents velocity component in subscript direction, V F is volume fraction of fluid in each cell, A i is fractional area open to flow in subscript

direction, p is pressure, ρ is fluid density, t is time, g i is gravitational force in subscript direction, f i is Reynolds stresses requiring a turbulence model for closure, V F and A i (cell face areas) = 1, thereby reducing the equations to the basic incompressible RANS equations. The Volume of Fluid (VOF) method is employed in Flow-3D. It consists of three main components: the definition of the volume of fluid function, a method to solve the VOF transport equation and setting the boundary conditions at the free surface. Turbulence is an important parameter hence it cannot be ignored in the simulations. Flow-3D can simulate the turbulence process by using mass and momentum conservation equations. In Flow-3D, there are five turbulence models available: the Prandtl mixing length model, the one-equation, the two-equation k-ε and Renormalized Group (RNG) models, and a large eddy simulation, LES, model (Flow Inc., User Manual v10). The RNG approach applies statistical methods to the derivation of the averaged equations for turbulence quantities, such as turbulent kinetic energy and its dissipation rate. It uses equations similar to the equations for the k-ε model. However, equation constants that are found empirically in the standard k-ε model are derived explicitly in the RNG model. Generally, the RNG model has wider applicability than the standard k-ε model. RNG is commonly used for the simulations of weir problems (Flow Inc., User Manual v10). 3.2 Model setup The simulations described in this study are run using version 10.0 of Flow-3D software on a PC.The geometries of both weir structures are created by using Flow-3D's solid modeler. One solid component with 6 subcomponents were created using simple box toolbar in the software (Figure 3). For turbulence-viscosity options, RNG (for all simulation cases) was selected. In addition, LES was selected in only one simulation case for the turbulence model comparison. The gravitational force was taken along the (–) z-direction and after defining the fluid type from the "Materials" tool, water was chosen as the fluid type with its properties predefined within the software. 3.3 Solution domain and meshing A computational mesh effectively discretizes the physical space. Each fluid parameter is represented in a mesh by an array of values at discrete points. Since the actual physical parameters vary continuously in space, a mesh with a fine spacing between nodes provides a better representation than a coarser one. Meshing is an important step to solve the hydraulic systems in numerical modelling. Size of cells or total cell numbers is the options to define the mesh block in the Flow-3D software. For the scope of this study, a constant cell size is specified to the model in meshing system instead of the number of total cells. Uniform mesh sizes of $\Delta x = \Delta y = \Delta z = 4$ mm were used for the prescribed solution domain. Additionally, the effect of

Figure 3. General 3D views of solid models constructed for contracted sharp crested weir.

mesh size on simulation results and computation time were investigated by applying different mesh sizes. For this purpose, one uniform mesh block is defined with cell sizes of 10 mm and 6 mm, other than 4 mm. Hence, the effect of mesh size on the results can be analyzed. To define a solution domain, created mesh block was taken half of the whole body in y-direction so that solution time optimized with the help of symmetry as shown in Figure 4.

3.4 Initial and boundary conditions

Appropriate boundary conditions are to be set in

the simulations to have a realistic result. The soft

ware includes six different boundaries since the flow

domain is defined as a hexahedral Cartesian coordi

nates (Kermani & Barani 2014).

Given in Figure 5 initial and boundary conditions

for the problem can be summarized as follows,

• X-min is chosen as pressure (P) where water level

is the input.

• X-max is outflow (O)

• Y-min, Y-max and Z-min are wall (W)

• Z-max are symmetry (S)

4 NUMERICAL MODELLING

The contracted sharp crested weir geometry studied by

Aydın et al. (2011) was used as previously mentioned.
Figure 4. View of the solution domains taken as symmetry of solid model. Figure 5. View of the boundary conditions on the solid model.

Table 1. Summary of model test parameters affecting

numerical simulation results.

Model parameter

tested Notes

Mesh cell size 4, 6 and 10 mm uniform mesh cell sizes are tested.

Turbulence model RNG and LES turbulence models are used within and outside the experimental data ranges.

Upstream channel 1, 1.5, and 2.5 m upstream channel length lengths are tested.

To obtain the optimum simulation model conditions for this geometry, the effects of

- mesh cell size
- turbulence model
- upstream channel length

were studied separately on a particular case with a defined upstream water depth.Then, after deciding the optimum model parameters with the specified physical definitions, the model was solved for the other water levels. The summary of the simulations is given in Table 1.

In order to determine the accuracy of the simulation results, the Relative Error (RE) between experimental and numerical discharge results are calculated using the Equation (9) given below

where Q cal is the discharge value calculated from the experimental study and Q CFD is the discharge value obtained from the simulation result. It should be clari fied that Q cal values are calculated by using the derived equations (Equations 2-6) byAydin et al. (2011). They are not the experimentally measured discharges.

For this study, only one weir width value, b = 0.16 m, is used, hence the corresponding b/B = 0.5 where the width of the channel is 0.32 m. The experimental data range of the water level above the weir crest is between $h_{min} = 0.0256$ m and $h_{max} = 0.2730$ m. For the given water levels, the corresponding discharges are $Q_{min} = 0.00134\ m^3/s$ and $Q_{max} = 0.04348\ m^3/s$, respectively. In this numerical study, the studied range includes five data points within the experimental range of Aydın et al. (2011) and three data points are outside the range of h. h_t is the total depth of flow in the channel. The weir height, P = 0.10 m. The values are provided in Table 2.

The results of the numerical study are compared to that of the empirical formulations given byAydın et al. (2011) which are given in Section 2 between Equations (2) and (6).

4.1 The effect of mesh cell size

Determining the appropriate mesh size is an important part of the numerical simulation. Grid size can affect not only the accuracy of the result, but also simulation time. The mesh with fine spacing provides better representation to the reality than the coarser one. However, refining the mesh means more computational time and it increases the size of numerical model. In other words, when the mesh size becomes smaller, the total mesh number increases, but increasing mesh number gives better results. Consequently, it is important to minimize the number of grids while including enough and sufficient resolution flow details (Kermani & Barani 2014; Flow Inc., User Manual, v10). In this part of the study, different mesh sizes with 10, 6, and 4 mm are

Table 2. Summary of the water levels used in the present study.

Table 3. Mesh cell size effect on simulation results for $h = 0.15$ m, $Q_{cal} = 0.01741\ m^3/s$.

Mesh size (mm)	Q_{CFD} (m^3/s)	RE (%)	Simulation Time (hrs)
10	0.01956	-12.35	0.26
6	0.01808	-3.85	2.33
4	0.01740	0.06	32.00

used to compare mesh size effect on RE of the results and simulation time. For this comparison, simulations were performed at the same water depth, which is h = 0.15 m.As an input, cell size option is used to define the mesh system instead of taking number of total cells. After determining the effect, the decision of the appropriate mesh size is made. As expected, mesh size of 10 mm has the least simulation time and mesh size of 4 mm has the longest simulation time. However, the simulation with mesh size of 4 mm gives the best results with the lowest RE value. Eventually, 4 mm for the mesh size is chosen for the other simulations. The results of these simulations are represented in Table 3. 4.2 The effect of turbulence model In this part of the study, LES and RNG turbulence models are used in the numerical simulations. Totally, four simulations are performed to find the effect of the turbulence model on simulation results. The two of them are of RNG model and the other two are of LES model. The reason of applying two simulations for each one

Table 4. Turbulence model effect for the data within the experimental range, h = 0.15 m, Q_{cal} = 0.01741 m^3/s.

Turbulence Model	Q_{CFD} (m^3/s)	RE (%)	Simulation Time (hrs)
RNG	0.01740	0.057	32.11
LES	0.01740	0.057	32.11

Table 5. Turbulence model effect for the data outside the experimental range, h = 0.30 m, Q_{cal} = 0.04977 m^3/s.

Turbulence Model	Q_{CFD} (m^3/s)	RE (%)	Simulation Time (hrs)
RNG	0.04988	-0.221	54.44
LES	0.05022	-0.904	53.72

Table 6. Effect of upstream channel length on simulation results for h = 0.15 m, Q_{cal} = 0.01741 m^3/s.

Upstream Simulation

channel length (m)	Q CFD (m^3/s)	RE (%)	Time (hrs)
2.5	0.01740	0.057	32.11
1.5	0.01768	-1.551	16.25
1.0	0.01770	-1.666	10.34

instead of single simulation for both cases is related to the experimental range. For each turbulence model, one simulation is done within the limit of experimental range conducted and the other is done for the outside of the experimental range. The results of the two dif ferent turbulence models are given in Table 4 and 5, respectively.

Although a slightly better RE value was obtained from RNG turbulence model in Table 5, one can state that two turbulence models give almost the same results. The reason might be due to the fact that tur bulence effect in the subcritical approach flow of the rectangular sharp-crested weirs is of little to no impor tance. However, in the present study, the RNG model is preferred as the turbulence model for other simula tion cases as it is known from Hargreaves et al. (2007) that RNG model has known advantages when there is strong curvature in the streamlines as in the case with accelerating flow over a weir.

4.3 The effect of upstream channel length

In this part of the study, three different upstream chan

nel lengths of 1.0, 1.5 and 2.5 m are used to predict the

effect of the channel length on simulation results. The

results of these simulations are represented in Table 6. Table 7. Comparison of the experimental data with the numerical results. Comp. Sim. h Q cal Q CFD Time RE Time (m) (m 3 /s) (m 3 /s) (sec) (%) (hrs) 0.05 0.00336 0.00350 25 -4.167 45.33 0.10 0.00936 0.00994 25 -6.197 16.76 0.15 0.01741 0.01740 20 0.057 32.00 0.20 0.02706 0.02700 20 0.222 18.79 0.25 0.03796 0.03798 20 -0.053 34.78 0.30 0.04977 0.05022 25 -0.904 54.44 0.35 0.06220 0.06350 20 -2.090 48.64 0.45 0.08789 0.09220 20 -4.904 65.94 Figure 6. Comparison of the numerical simulations with experimental values. Figure 7. Variation of absolute value of RE with h. It is seen fromTable 6 that shorter upstream channel increases the RE between the experimental and numerical results. Therefore, upstream channel length of 2.5 m is accepted as sufficient to obtain accurate simulation results. The channel length may be lengthened further but in this case the number of computational cells would increase significantly and hence would increase dramatically the simulation time. 4.4 Numerical simulations using various upstream flow depths Using the optimized model parameters of mesh size, turbulence model, and upstream channel length which

Table 8. Comparison of the numerical results with the

results obtained from head-discharge relation.

h	Q	Q CFD	RE
(m)	(m 3 /s)	(m 3 /s)	(%)
0.05	0.00331	0.00350	0.0142
0.10	0.00932	0.00994	0.0047
0.15	0.01719	0.01740	0.0129
0.20	0.02663	0.02700	0.0158
0.25	0.03750	0.03798	0.0122
0.30	0.04968	0.05022	0.0018

0.35 0.06311 0.06350 -0.0146

0.45 0.09352 0.09220 -0.0641

were obtained from the studies given in the pre vious sections, systematical simulations have been performed for various upstream flow depths of the contracted weir studied. The experimental values of upstream flow depth data for b/B = 0.5 varied from 0.0256 m to 0.2730 m. For this constant b/B ratio, five flow depths within and three flow depths outside of the experimental range were given as initial bound ary condition in the numerical model. The comparison of numerical and experimental results is given both in Table 7 and in Figure 6, while the variation of absolute values of RE with the upstream flow depth, h is given in Figure 7. The results showed that the formulation given by Aydin et al. (2011) for discharge computation in contracted sharp crested weirs can be considered as valid both within and up to a certain limit for outside of the experimental range. Absolute values of the maxi mum and the minimum RE were obtained as 6.197 and 0.053%, respectively, indicating reasonably acceptable simulation results.

In order to validate the numerical results of the present study, discharges obtained are also compared with the well-known head-discharge relation given by Kindsvater and Carter (1957). The form of the proposed equation is similar to Equation (1) with modifications to weir width, b and head over the weir, h. The discharge coefficient, C_d is defined as a function of the ratio of b/B and h/P. The result of the comparison is given in Table 8. RE values are calculated by using Equation (9) where Q_{cal} is replaced by the discharge, Q that is calculated by using modified version of Equation (1). It can be seen from Table 8 that the values of RE are all very small indicating an acceptable simulation.

5 CONCLUSIONS

A numerical simulation study was performed by using a commercially available CFD software Flow-3D in order to analyze flow characteristics for sharp crested contracted weir which is a commonly used discharge measurement structure. Firstly, the optimization of numerical model parameters such as mesh size, turbulence model type, and upstream channel length were

Hirt, C. W. & Nichols, B. D. 1981. Volume of fluid (VOF) method for the dynamics of free boundaries. Journal of Computational Physics 39(1), 201–225.

Kermani, E. & Barani, G. A. 2014. Numerical simulation of flow over spillway based on the CFD method. Journal of Scientica Iranica A 21(1): 91–97.

Kindsvater C. E. & Carter, R. W. 1957. Discharge character

istics of rectangular thin-plate weirs. Journal of Hydraulic

Division ASCE (83): 1–36.

Ozmen-Cagatay, H. & Kocaman, S. 2010. Dam-break flows

during initial stage using SWE and RANS approaches.

Journal of Hydraulic Research 48(5): 603–611.

Qu, J. Ramamurthy, A. S. Tadayon, R. & Chen, Z. 2009.

Numerical simulation of sharp-crested weirs flows. Cana

dian Journal of Civil Engineering 36: 1530–1534.

Rady, R.A. 2011. 2D-3D Modeling of flow over sharp-crested

weirs. Journal of Applied Sciences 7(12): 2495–2505. Ramamurthy, A. S. Tim, U. S. & Rao, M. V. 1987. Flow over sharp crested plate weirs. Journal of Irrigation and Drainage Engineering ASCE 113(2): 163–72. Rehbock, T. 1929. Discussion of precise weir measurements by EW Schoder and KB Turner. Trans. ASCE. 93. 1143– 1162. Samadi, A. & Arvanaghi, H. 2014. CFD simulation of flow over compacted compound arched rectangular sharp crested weirs. International Journal of Optimization in Civil Engineering 4(4): 549–560. Swamee, P. K. 1988. Generalized rectangular weir equations. Journal of Hydraulic Engineering ASCE 114(8):945–9. S¸is¸man, Ç. 2009. Experimental investigation on sharp crested rectangular weirs. M.S.Thesis submitted to Middle East Technical University. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Field measurements at the new lock of Lanaye (Belgium) before the opening to navigation

C. Savary, D. Bousmar, C. Swartenbroekx & G. Zorzan

Hydraulic Research Laboratory, Service Public de Wallonie, Châtelet, Belgium

T. Auguste

Engineering Structural Expertise, Service Public de Wallonie, Liège, Belgium

ABSTRACT: A new navigation lock (length: 225 m, width: 25 m, drop: 13.7 m) was inaugurated in 2015 in

Lanaye in Belgium. Before opening the lock to navigation, field measurements were realized to check its good

working and to compare the operating conditions with the acceptable thresholds corresponding to the design

criterions. The measurements during the lockage allowed to control: (i) the filling and emptying hydrographs, (ii)

the movement of water in the lock chamber, (iii) the wave amplitude in the downstream and upstream reaches,

(iv) the pressure, the vibrations and the sound level around the valves in order to quantify the cavitation level

and (v) the vibrations at different places of the structure. In an attempt to find a solution to decrease vibration

and cavitation level noticed during the trial period, the hydraulic measurements were realized for two opening

laws of the valves.

1 INTRODUCTION

In the frame of the 18th priority project of theTransEu

ropean Network of Transport (TEN-T) concerning the

corridor Rhine-Main-Danube, a new navigation lock

was inaugurated in 2015 in Lanaye on the borderline

between Belgium and The Netherlands. The new lock

is 225 m long and 25 m wide and will allow vessels up

to 9000 tons (CEMT Class VIb) to cross a difference

in height of 13.7 m between the Albert canal and the

Lanaye canal.

The filling and emptying system of the new lock is composed of longitudinal culverts with side ports, the filling and emptying discharge is regulated with butter fly valves of 3.5 m diameter. The geometry of this sys tem was designed in collaboration with UCL (Univer sité catholique de Louvain) using the numerical model ALFREDO (Debources & Zech 1996, Escarmelle & Zech 1998 and Vancaster & Zech, 1999). An opti mization was made on the geometry regarding dif ferent criterions: (i) time for a leveling < 20 minutes; (ii) free-surface slope in the lock chamber < 0.5 % to limit the forces in the mooring lines; (iii) emptying dis charge not too high to limit waves in the downstream reach.

During the trial period, it was noticed that the struc tures (technical buildings and galleries) were subject to vibrations during the lockage and that cavitation occurred around the butterfly valves.

Before opening the lock to navigation, field mea surements during the lockage were realized with two main purposes. The first purpose was to check the good working of the lock and to compare the operating conditions with the acceptable thresholds corresponding to the design criterions. The second purpose was to quantify the cavitation and vibration level. In an attempt to find a solution to decrease this level, the measurements were realised for two opening laws of the valves: (A) linear 90 ∘ in 9 min as designed; and (B) bi-linear 40 ∘ in 1 min, 90 ∘ in 6 min. After a short description of the set up and the schedule of the measurements, the experimental results are presented and analyzed. For different lock operations (filling, emptying, with one or two culverts, for the two opening laws tested), the following results are discussed: (i) the filling and emptying hydrographs; (ii) the water slope in the lock chamber; (iii) the wave amplitude in the downstream and upstream reaches; (iv) the pressure, the vibrations and the sound level around the valves in order to quantify the cavitation level; and (v) the vibrations at different places of the structure. 2 MEASUREMENT SET UP 2.1 Water level and valve motion The location of the different gauges measuring the water level and the motion of the valves is illustrated in Figure 1.

Figure 1. New lock of Lanaye, view from above, location of the gauges.

Figure 2. Installation of the pressure gauges in the lock

chamber.

ZL1-5, ZR1-3, ZC1-3, Zup4 and Zd1 are abso

lute pressure gauges (Druck PMP4030) with a range

from 10 to 30 mH 2 O. ZL1-5 and ZR1-3 were fas

tened on some of the ladders in the lock chamber

(Fig. 2). The measurements of these gauges are used

to estimate the free-surface slope in the lock chamber.

ZC1-3 are slid down in the stop log recesses on the

left side (downstream of the upstream valve, upstream

and downstream of the downstream valve), they are

used to calculate the head loss at the valve and in the

culvert. Zup4 and Zd1 indicate the water level in the

upstream and downstream reaches. VupL, VupR, VdL

and VdR are cable position sensors (0–2 m) that mea

sure the extension of the hydraulic jack which activates

the butterfly valve. The angular position of the valve can be deduced. All these gauges are connected by wire to 3 data acquisition devices (UpL, UpR and DownL in Figure 1) located in the technical buildings. The acquisition rate is chosen at 20 Hz. Synchronization in time between the different data acquisition devices is insured by connecting them together with a cable loop on which a 10V pulse is given at the beginning of each measurement. A perfect synchronization in time is needed to deduce the free-surface slope from the individual water level measurements. A lack of synchronization of 1 s could lead to an error of 0.1 % on the slope which is quite important regarding the small value expected (<0.5 %). Zup1-3 and Zd2-5 are absolute pressure gauges with internal data logger (Diver DI501) with a range from 10 to 20 mH 2 O. They are used to measure the propagation of the waves in the reaches due to the lockages. The acquisition rate is 0.2 Hz. A perfect synchronization is not required. All these devices were synchronized before the beginning of the campaign. 2.2 Cavitation It is not possible to have an absolute measurement of the cavitation level around the valve. Nevertheless different indicators related to cavitation can be deduced from measurements and allow to compare different configurations together. Non-dimensional coefficients are proposed in literature (Tullis 1993, Baran et al. 2007, Bleuler, 1939) to estimate the cavitation level. These coefficients depend of the pressure upstream and downstream of the valve (measured respectively by ZC2 and ZC3 for the downstream valve). Implosion of the vapor pockets due to cavitation makes noise and vibration (high frequency). A hydrophone (TC 4013 Teledyne Reson, maximum frequency 100 kHz) is used to measure the sound pressure in water. It is placed outside the steel pipe downstream of the valve in acoustic gel to insure a good sound transmission with the pipe. A triaxial accelerometer (Vibrasens 138.01.6D2, sensitivity 100 mV/g, maximum frequency 10 kHz) is placed near the hydrophone

Figure 3. Installation of the hydrophone and the accelerom

eter on the valve.

(Fig. 3). The hydrophone and the accelerometer are

connected to a specific acquisition device. Due to

the important acquisition rate (hydrophone: 200 kHz,

accelerometer XYZ: 20 kHz), the measurements are

not realized continuously during the leveling: each

10 s, the measurements are recorded during 1 s. To analyze lower frequencies, a velocity sensor (500 Hz) recording measurements continuously was mounted on the valve. In addition, the noise level was recorded by a sonometer.

2.3 Vibrations

Accelerometers were dispatched at different places of the technical buildings and galleries around the upstream and downstream valve chamber. Two vibra tion measurement systems, each one composed of three velocity sensors, were disposed on the ground and the first floor of the control building. The sys tem was programmed to record signal greater than 0.2 mm/s, to record vibrations due to the lockage but not vibrations due to the work traffic.

3 MEASUREMENT SCHEDULE

The measurements took place during 5 days including 1 day for the installation of the equipment and 1 / 2 day for the dismantling. There was no navigation through the new lock during the tests.

In an attempt to decrease the vibration level and the duration of cavitation, an alternative law was tested for the opening of the valve. Figure 4 illustrates the law recommended from the design (low A, linear open ing) and the alternative law corresponding to a faster opening of the valve at the beginning (law B, bi

linear opening). The alternative law was chosen on the

basis of a preliminary numerical study concerning the

limitation of the duration of cavitation (Savary & Lib

ert 2013). In this study, the lockage was numerically

modeled with ALFREDO (Christiaens et al. 2014,

Christiaens & Soares-Frazão 2013). Figure 4. Tested opening laws of the valves. Table 1. Configurations for the tests. Valve opening law A law B Filling Symmetrical FSA1-4 FSB1-3 Asymmetrical FAA1-2 FAB1-2 Emptying Symmetrical ESA1-6 ESB1-4 Asymmetrical EAA1-4 EAB1-2 For each valve opening law, the measurements were realized for several configurations of the lockage: emptying, filling, symmetrical (two culverts), asymmetrical (one culvert). Moreover, in order to check the reproducibility, several measurements were realized for each configuration. The denomination of the lockage for which measurements were made, used in this paper, is given in Table 1. A waiting time (≈20 min) is necessary between two tests to have a smooth free-surface in the lock chamber at the beginning of each test. 4 HYDROGRAPHS The data are recorded at the rate of 20 Hz, a sliding average is made over 1 second. To limit the amount of data, only 1 measurement per second is considered. The filling or emptying discharge Q can be calculated from Equation 1. where z = mean water level in the lock chamber and A = surface of the lock chamber. A good reproducibility is observed between the measurements corresponding to the same configuration. The evolution with time of the mean water level in the lock chamber and the discharge is plotted in Figure 5 for a symmetrical emptying of the lock for the opening laws A and B of the valve. The notation DNG expresses that the reference level is the Belgian standard level of the sea. Table 2 summarizes the maximum discharge and the duration of the lockage for the different tested configurations. As the upstream and downstream levels

Figure 5. Evolution of the discharge and the water level in

the lock chamber during a symmetrical emptying with the

two tested opening laws (ESB1 and ESA1).

Table 2. Maximum discharge and duration of the lockage.

			Maximum discharge (m^3/s)	Duration of the lockage
LawA	Filling	Sym.	160	16 min 45 s
		Asym.	95	28 min 30 s
	Emptying	Sym.	153	17 min 40 s
		Asym.	92	28 min 30 s
Law B	Filling	Sym.	163	13 min 30 s
		Asym.	98	24 min 40 s
	Emptying	Sym.	156	14 min 30 s
		Asym.	92	25 min 40 s

were not exactly the same between the different tests, for a same configuration, there are small differences between the results. The results presented in Table 2 correspond to the mean values.

The maximum discharge corresponding to the opening law B of the valve is slightly (≈2%) larger than for the law A and the duration of the lockage is about 3 minutes smaller.

The measured maximum discharge for a symmetrical emptying of the lock (law A) is more or less 5% larger than the maximum discharge calculated with the numerical model for the design of the lock (Vancaster & Zech 1999). In this case, the duration of the lockage was overestimated by the numerical model of around 50 s. These differences may be due to small modifications in the culverts layout between the initial and final projects, but also to some inaccuracies in head loss coefficients estimation.

5 FREE-SURFACE SLOPE IN THE LOCK CHAMBER

In order to limit the forces in the mooring lines, the slope of the free-surface in the lock chamber during

the lockage has to remain below 0.5% for all kind

of vessels. This criterion is conservative compared to Figure 6. Example of a linear regression to calculate the mean slope. Figure 7. Evolution of the mean water slope in the lock chamber during a symmetrical fillng with the two tested opening laws. admissible forces as listed by PIANC (2015): longer is the ship, larger is the limit slope, the smallest limit slope is 0.85% for a Va CEMT vessel. This conservative criterion accounts for considering the water slope instead of the actual force. However, in situ measurements on other locks revealed larger water-surface slopes, which do not imply problem for vessels during the lockage. The mean slope of the free-surface (representative for long vessels) is estimated on basis of a linear regression taking into account all the water level measurements in the lock chamber (ZL1-5 and ZR1-3). The mean slope of the water is assumed to be the slope of the linear regression. An example is illustrated in Figure 6 where the measurements 5 min after the beginning of the lockage are reproduced for FSA4. Regarding the slope of the water-surface, the filling is more critical than the emptying of the lock and an asymmetrical lockage is more critical than a symmetrical one. Figure 7, illustrates the evolution of the mean freesurface slope in case of a symmetrical filling of the lock with the two tested opening laws of the valves. The slope is larger with the opening law B, but in all cases, the mean slope remains below 0.5%.

Table 3. Position of the pressure gauges.

Upstream Downstream

Albert canal Lanaye canal Maas

Zup1 Zup2 Zup3 Zd2 Zd3 Zd4 Zd5

km km

1 0.5 0.05 0.05 0.5 1.2 3.2

The maximum longitudinal slope and transver

sal slope (maximum difference between two nearby

gauges) in case of a symmetrical filling remain lower than 1%.The transversal slope exceeds sometimes 5% in case of an asymmetrical filling. Safety guidelines have to be recommended in case of an asymmetrical filling (i.e.: only one boat on the width of the lock chamber moored on the side of the operational valve). The extreme slopes are similar, but a little larger for the opening law B in comparison with the law A.

6 WAVES DUE TO THE LOCKAGE IN THE UPSTREAM AND DOWNSTREAM REACHES

Pressure gauges were placed in the upstream (Zup1-3) and downstream (Zd2-5) reaches to measure the wave propagation due to the lockages (position in Table 3). The new lock, named Lanaye 4, is the 4th lock built on the site of Lanaye (Lanaye 1 and 2 are 7.5 m × 55 m and Lanaye 3 136 m × 16 m). To do the measurements in optimal conditions, the navigation should have been stopped, but it was not the case. The waves due to the lockage of the Lanaye 3 lock (16 m × 136 m) and the wakes due to navigation were also recorded.

Figure 8 illustrates the waves measured in the downstream reach due to a simultaneous emptying of the locks Lanaye 3 and Lanaye 4 and the emptying hydrographs of these two locks. For a better visualiza tion, the curves corresponding to the measurements of Zd3, Zd4, Zd5 were each shifted of 0.5 m on the ordinate axis.

In this case (the most critical one), the maximal amplitude of the wave at Zd2 (outlet of the lock) reaches 55 cm which is reduced to 25 cm at Zd4 (extremity of the downstream guide wall), in the Maas (Zd5), the wave is lightened. The smallest measured waves correspond to a case for which the lockage of Lanaye 4 stops when the emptying discharge of Lanaye 3 is maximal. In this case the maximal amplitude of the wave is 25 cm in Zd2 and 11 cm in Zd4. No difference is noticeable between the opening laws A and B. The natural period corresponding to the Lanaye canal (time between two wave peaks) is 21 min.

In situ measurements were realized in 1995 (Bertrand et al. 1995) to quantify the waves due to the lock of Lanaye 3.At that time, the maximum amplitude

measured was 48 cm at the outlet of the lock Lanaye 3

Figure 8. Time evolution of the waves in the downstream reaches and emptying discharge from the locks Lanaye 3 and 4.

and 23 cm at Zd4, the natural period of the canal was 18 min. The situation before and after the works of the new lock can be compared. In the worst configuration (simultaneous lockage of Lanaye 3 and 4), the amplitude of the wave is 10% larger in the present configuration. Nevertheless, when the emptying of the two locks is not in phase, the amplitude is smaller than before the construction of the new lock. Moreover, due to the construction of the new lock, the downstream port was enlarged which increased the period of the Lanaye canal. This increase implies a smaller slope and velocity variation. The waves in the upstream reach due to a filling of the locks are of lower amplitude. The amplitude is between 10 cm and 20 cm at Zup1 depending on the synchronization between the locks (Lanaye 3 and 4) and the smoothness of the water level surface at the beginning of the lockage. Moreover the wave period is longer (a reflection of the wave is not measured). 7 CAVITATION AROUND THE BUTTERFLY VALVES Cavitation is characterized by the formation of vapor in the flow due to a local drop of pressure under the vaporization threshold at ambient temperature. Such a pressure drop occurs when the velocity is locally high, for example when the flow is contracted around a butterfly valve. The level of cavitation is estimated regarding the cavitation coefficients and the measurements of noise and vibrations. 7.1 Cavitation non-dimensional coefficients Several non-dimensional coefficients are used to characterize the cavitation (Tullis 1993, Baran et al. 2007, Bleuler 1939). They express the ratio between the physical parameters that limit the cavitation (upstream

Figure 9. Evolution of the cavitation coefficient σ and the cavitation threshold values during a symmetrical emptying (law A).

or downstream pressure) and those that increase cavi tation (the head loss and/or the velocity). Larger is the non-dimensional coefficient, lower is the cavitation.

The non-dimensional cavitation coefficients σ and C used in the following are:

where H_1 and H_2 = absolute piezometric level upstream and downstream of the valve, respectively, (reference = centre of the pipe), H_{vap} = vaporization pressure (at 20 ◦ C, $H_{vap} \approx 0.3$ m H_2O), $\Delta H = H_1 - H_2$ = difference in piezometric level at the valve and v = mean velocity in the pipe downstream of the valve.

The evolution of the cavitation coefficient σ (Eq.2) around the downstream valves during a symmetrical emptying with the opening law of the valves A is plotted in Figure 9.

Tullis (1993) proposed threshold values for σ (Eq.2) corresponding to different stages of cavitation. These threshold values are plotted in dotted line in Figure 9, from the lightest (incipient cavitation, top curve) to the most severe level of cavitation (maximum cavitation, bottom curve). These threshold values are specific to butterfly valves and vary with the opening angle of the valve. They were obtained experimentally by Tullis (1993). Scale effect coefficients were applied to take into account the difference in size and pressure con dition between the valves of the lock and the valves used at the laboratory by Tullis (1993). Nevertheless, the absolute value obtained for these thresholds have to be used with care because the size scale effect was never validated on valves with such diameter (study limited to 1 m). Figure 9 illustrates that an intense cav itation level is reached during the emptying of the lock. The cavitation level exceeded the incipient damage

threshold during 7 min 30 s. Figure 10. Evolution of the cavitation coefficient C during a symmetrical and asymmetrical emptying (law A). Figure 11. Evolution of the cavitation coefficient C during a symmetrical emptying for the opening laws A and B. Figure 10 compares the evolution of the cavitation coefficient C (Eq.3) for a symmetrical and an asymmetrical emptying with the law A. Concerning the non-dimensional cavitation coefficient C (Eq. 3), Bleuler (1939) proposed, on basis of experimental measurements, a threshold value C = 2.5 corresponding to the beginning of cavitation for butterfly valves. According to Bleuler

(1939), this threshold value does not change with the opening angle.Another threshold value C = 1 corresponding to intense cavitation is mentioned in literature (i.e.: Kurkjian & Pratt 1974). Figure 10 shows that the cavitation last a longer time for an asymmetrical emptying, it is also a little stronger. Figure 11 compares the evolution of the cavitation coefficient C (Eq.2) for a symmetrical emptying of the lock with the 2 tested valve opening laws. The level of cavitation reached is similar in both configurations, but the duration of cavitation is 3 minutes smaller with the opening law B. The upstream valves are also subject, in a similar way, to an important level of cavitation during the filling of the lock. Table 4 summarizes the duration of cavitation for the different measured lockages (t C≈1 and t C<2.5 are

Table 4. Duration of cavitation, non-dimensional coeffi

cients σ and C. t C≈1 t C<2.5 t σ<id

LawA Filling Sym. 3 min 8 min 7 min 30 s Asym. 4 min 10 min 20 s – Empt. Sym. 2 min 30s 8 min 7 min 30 s Asym. 3 min 30 s 11 min –

Law B Filling Sym. 40s 4 min 30 s 4 min 30 s Asym. 1 min 8 min – Empt. Sym. 10 s 4 min 30 s 4 min 30 s Asym. 1 min 30 s 7 min –

the durations corresponding to cavitation coefficient

C ≈ 1 and C < 2.5, t σ<id is the duration corresponding

to cavitation coefficient σ smaller than the incipient

damage threshold).

7.2 Sound and vibration

The signal of the hydrophone and the accelerometer

located on the valve is recorded during 1 s each 10 s.

In order to know the evolution of the mean intensity of

the signal in time, for each package of measurements

of 1 s, we calculated the root mean square (RMS).

where x = measurement (in Pa for the hydrophone and in g for the accelerometer) and n = number of measure

ments in 1 s (200,000 for the hydrophone and 20,000

for the accelerometer).

To have an idea of the peak values, an indicator,

related to the RMS is calculated.

The evolution of the RMS and the peakvalue during

a symmetrical emptying with the law A is plotted in

Figure 12 for the measurements of the hydrophone.

The RMS reaches a maximum value between 1 min

and 5 min 30 s after the opening of the valve. Then

the noise decreases to become negligible after 12 min.

Between 6 and 8 min, the noise level decreases, but the

peakvalue is larger. It may correspond to a still existing

cavitation but less continuous than earlier.

The RMS values from the measurements of the

hydrophone are plotted in Figure 13 for different

emptying configurations.

The duration corresponding to the maximal noise

level is smaller for the opening law B of the valve.

The noise decreases more slowly when the emptying

is asymmetrical.

Figure 14 illustrates that the RMS of the noise level

decreases quickly when the cavitation coefficient C Figure 12. Evolution of the RMS and peakvalue (from the measurements of the hydrophone) during a symmetrical emptying (law A). Figure 13. Evolution of the RMS (from the measurements of the hydrophone) during the emptying of the lock for the different tested configurations. Figure 14.

Evolution of the RMS (from the measurements of the hydrophone) with the cavitation coefficient C. increases. The relation between these two parameters is similar for the different emptying configurations. Figure 15 compares the RMS calculated with the measurements of the hydrophone and the accelerometer. There is a strong linear correlation between the noise level and the vibration.

Figure 15. Correlation between the RMS calculated

from the measurements of the hydrophone and of the

accelerometer.

Table 5. Duration of intense and moderate noise and vibra

tion level. Noise and vibration level intense moderate

LawA Filling Sym. 2 min 5 min 30 s Empt. Sym. 4 min 30 s 5 min Asym. 4 min 30 s 11 min

Law B Filling Sym. 30 s 4 min Empt. Sym. 2 min 30 s 4 min 30 s Asym. 3 min 30 s 9 min 30 s

A spectral analysis of the measurements was real

ized. It revealed that the signal was distributed among

different frequencies, none specific frequency was

highlighted.

Table 5 summarizes the duration of intense and

moderated noise and vibration level for the different

configuration tested.

In Table 5, intense sound and vibration level corre

sponds to RMS value of the sound pressure > 100 Pa

and RMS value of the acceleration > 1.5 g. Mod

erate sound and vibration level corresponds to

100 Pa > RMS value of the sound pressure > 10 Pa

and 1.5 g > RMS value of the acceleration > 0.2 g.

The maximum sound level recorded by the sonome

ter located near the dowstream valve was 100 dB.

8 VIBRATION OF THE STRUCTURE

The vibrations in the structures were essentially notice

able at the first floor of the control building (located

near the downstream door of the lock, DownL on Fig

ure 1) where large (not rigid enough) glass walls easily

vibrated.

The amplitude of the vibration measured by the

velocity sensors in this building (<0.7 mm/s) was

largely below the limit for building degradation

(5 mm/s). Figure 16. Time evolution of the spectrum coming from the pressure measurements ZC2. Two periods of vibrations were identified. (i) The first one occurred for an opening of the valve around 50 ∘ . Several frequencies were concerned with limited amplitude. It occurred for all emptying operations. (ii) The second period of vibration was only noticed in case of asymmetrical emptying. It happened when the valve was completely open, close to the maximum discharge. These vibrations had a larger amplitude and were characterised by a sine wave around 6 Hz lasting approximately 5 min. They were measured by the three sensors located at the first floor of the control building. These two periods of vibrations were also noticeable in the water pressure measurements in the culverts (ZC1, ZC2 and ZC3). Figure 16 represents the result of the spectral analysis applied on the measurements of ZC2 during an asymmetrical emptying with the opening law A. To observe a time evolution, the signal was divided into periods of 10 s. For each period, a spectral analysis was made on the measurements giving the power spectrum density (colored scale) as a function of the frequency. In Figure 16, two areas of pressure fluctuations correspond to the vibrations recorded at the first floor of the control building. The duration and the intensity of the vibration were more or less the same for the two tested opening law of the valves. The periods of vibration took place earlier with the law B because the vibrations occured for the same opening angle for both laws. Concerning the measurements of the seven

accelerometers dispatched on the wall of the valve chamber and the technical galleries, the predominant frequencies measured at each accelerometer were different. The frequency of 6 Hz was never measured as a predominant frequency. 9 CONCLUSION Different measurements were realized on site to check the conformity of the lock with the hydraulic design criterions. Two opening laws of the valves were tested in order to decrease the cavitation and vibration problems.

The maximum discharge is 5% larger than the discharge calculated during the design study and the duration of the lockage is a little smaller.

Regarding the slope of the water surface in the lock chamber, the mean slope remains below the conservative value of 0.5 %. Nevertheless the transversal slope for an asymmetrical filling reaches 5 %. For this case, operational guidelines have to be proposed to avoid safety problems. The slopes measured for the opening law B are slightly larger than for the opening law A.

The waves in the upstream reach due to the filling of the new lock are less critical than the waves in the Lanaye canal downstream of the lock. When the locks Lanaye 3 and 4 are emptying in phase, the amplitude of the downstream wave is 10% larger than before the construction of the new lock. In other cases, the amplitude is lower, partially thanks to the widening of the flume to build the downstream port of the new lock.

An intense level of cavitation is reached around the upstream and downstream butterfly valves. The open ing law B allows decreasing the duration of cavitation (≈3 min) against the law A. To have a better idea of the possible damages associated to cavitation, an inspec tion of the valve has to be planned after one year of use.

Cavitation and vibration of the structures are two different problems. The measured vibrations are not dangerous for the structure. Nevertheless, a vibration having a specific frequency of 6 Hz was measured at the first floor of the control building in case of asym metrical emptying. Pressure waves corresponding to this frequency have also been measured in the culverts. The origin of this pressure wave still has to be inves tigated. The vibration level and amplitude are similar for the two tested opening laws.

ACKNOWLEDGMENT

The authors acknowledge the contribution of the tech nical staff of the hydraulic research laboratory and of the engineering structural expertise department for their participation to the preparation and realization of the experimental campaign. Vincent Herbillon and Philippe Fransolet from the waterways department of Liège and the pilots of the dinghy, Christian Muyshond and Jules Staes from the waterways department of Charleroi are also thanked for their contribution dur ing the tests. Finally, the department of hydrology is acknowledged for the lending of some materials and the transmission of hydrological data.

Bertrand, G., Zorzan G. & Hiver J-M 1995. Quatrième écluse de Lanaye, Etude des ondes de sassement – rapport intermédiaire 1, Campagne de mesures dans le bief aval du 23 et 24-25 septembre 1995. Laboratoire des Recherches hydrauliques, M.E.T., Chatelet, Belgique. (in French) Baran, G., Bunea F.D & Oprina, G.I. 2007. On cavitation and cavitational damage at butterfly valves. 2nd IAHR International Meeting of theWorkgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems. Timisoara, Romania, October 2007 Bleuler, H. 1939. L'écoulement avec cavitation dans les vannes-papillon. Bulletin Escher Wyss, tome 12, n ∘ 1-2, 31-35. S.A. des Ateliers de Constructions Mécaniques Escher Wyss, Suisse. (in French) Christiaens D., Bousmar D. & Soares-Frazão, S. 2014. Simplified numerical model of a navigation lock levelling with a ship in the lock chamber. PIANC Congress San Francisco, USA 2014. Christiaens, D. & Soares-Frazão, S. 2011-2012-2013. Écluse 225 m X 25 m d'Ampsin-Neuville, Assistance à l'étude par modélisation physique et numérique du système d'alimentation en eau du sas. Rapports intermédiaires rédigés dans le cadre du marché de service MS/222/11/01 pour le compte de la Direction des Recherches Hydrauliques du SPW par le pôle de Génie Civil et Environnemental, Université catholique de Louvain, Belgique. (in French) Debources, C. & Zech, Y. 1996. Etude comparative de systèmes de remplissage d'une écluse. Rapport rédigé dans le cadre de la convention MS/213/95/1 pour le compte du Laboratoire de Recherches Hydrauliques du MET par le Laboratoire de Génie Civil, Université catholique de Louvain, Belgique. (in French) Escarmelle, A. & Zech Y. 1998. Utilisation de modèles numériques de simulation de sassement d'une écluse de navigation fluviale. Rapport rédigé dans le cadre de la commande 97/2455/X.6.2.5.2 pour le compte du Laboratoire de Recherches Hydrauliques du MET par le Laboratoire de Génie Civil, Université catholique de Louvain, Belgique. (in French) Kurkjian G.A.Jr & Pratt H.C. 1974. Follow these simple rules to get long life from your large butterfly valves. Air Gas and Liquid Handling, Power, July 1974, McGraw-Hill, Inc. Savary, C. & Libert, Y. 2013. Quatrième écluse de Lanaye, système de remplissage et de vidange, evaluation du risqué de cavitation au droit des vanes de sassement. Rapport final, SPW, Châtelet, Belgium. (in French) PIANC 2015. Ship behavior in locks and lock approaches. Report 155 Tullis, J.P. 1993. Cavitation Guide for Control Valves. NUREG/CR-6031. Prepared for the US Nuclear Regulatory Commission. Vancaster P-H. & Zech, Y. 1999. Utilisation de modèles numériques de simulation de sassement d'une écluse fluviale. Rapport rédigé dans le cadre de la convention MS/213/96/2 pour le compte du Laboratoire de Recherches Hydrauliques du MET par le Laboratoire de Génie Civil, Université catholique de Louvain, Belgique. (in French) Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Experimental investigation of the influence of breaking logs on the flow patterns induced by lock filling with gate openings

K. Verelst & J. Vercruysse

Flanders Hydraulics Research, Antwerp, Belgium

P.X. Ramos & T. De Mulder

Hydraulics Laboratory, Civil Engineering Department, Ghent University, Ghent, Belgium

ABSTRACT: During the design of a filling and emptying system of a lock with openings in the lock gate, inserting breaking logs at the downstream side of the lock gate is one of the potential means to enhance the energy dissipation of the filling jets and to reduce the hydrodynamic forces on the ships moored in the lock chamber. To investigate the influence of breaking logs on flow patterns and energy dissipation in the lock chamber, a dedicated (generic) physical model was built, consisting of one circular opening in a lock gate sealed by a vertical lift valve. Both configurations without and with breaking logs, having a square cross-section, were tested. The configurations with breaking logs differ with respect to

the spacing and the number of the breaking logs. First, the influence of the breaking logs and the associated blockage of the opening by the breaking logs onto the discharge coefficient of the opening was assessed. The velocity measurements and the visualisation of the flow pattern revealed the spreading of the jet and the corresponding velocity decay at short distances downstream of the gate opening. Also an asymmetrical behaviour of the jet and backflow effects were noticed, due to the dimensions and downstream boundary conditions of the model.

1 INTRODUCTION

Navigation locks are key structures for the accessibil ity of ports and for navigation in canals and canalized rivers. Different systems exist for filling (and empty ing) the lock chamber. For relatively high-head locks mostly short culverts bypassing the lock gates (with a stilling chamber), sidewall filling systems (with lon gitudinal culverts and ports) or bottom filling systems are considered. In Belgium, the head for most inland navigation locks, however, is limited to 2–3 m. For such low-head locks, through the gate filling systems are usually considered, i.e. lock filling takes place through openings in the lock gate (often mitre gates) sealed by some valves (often vertical lift valves). The latter levelling system is the subject of the research in this paper.

During recent designs of filling and emptying sys tems with openings in mitre gates in Belgium, for constructive reasons, circular openings in the gate are considered. These circular openings consist typically of a circular tube with a length equal to half the width of the lock gate. The circular tube is situated within a rectangular chamber, consisted of the horizontal and vertical beams the lock gate is constructed with. Con sequently the width and height of this chamber is larger than the diameter of the circular opening and allows the widening of the jet towards the downstream end of

Figure 1. the gate opening. Figure 1 presents a sketch of a lock gate with a typical circular opening in it. In the hydraulic design of a levelling system with openings in the lock gate, a compromise should be found between an acceptable lock filling time and acceptable forces experienced by the moored vessels during the process. One of the potential means to improve the energy dissipation of the filling jets and reduce the hydrodynamic forces on the ships, is to insert breaking logs (also referred to as energy dissipation bars) at the downstream side of the gate openings. These breaking logs are intended to enhance the spreading and energy dissipation of the filling jets.

In this paper an account will be given of research carried out in a dedicated lab facility, in order to investigate the influence of breaking logs on the induced flow patterns and their role in the energy dis sipation of a turbulent filling jet. However, to reduce the complexity, the physical model considers only a gate perpendicular to the lock chamber's longitudi nal axis and only one circular opening in the gate sealed by a vertical lifting valve. Consequently only one filling jet is entering the lock chamber, while in reality more than one opening is present in the lock gate. Also no vessels are present in the lock chamber. The physical model tests described in this paper also only consider breaking logs with a square section, which are positioned vertically. Other geome tries (horizontal position of breaking logs, I-profile breaking logs), already tested or planned to be tested in the physical model, are not considered in this paper.

The outline of the paper is as follows. Section 2 describes the physical model and the measurement techniques used. The influence of breaking logs on the discharge coefficient of the opening in the lock gate is discussed in section 3. Section 4 presents the visuali sation of the flow pattern downstream of the lock gate, while section 5 discusses the flow velocity measure ments downstream of the lock gate. Section 6 provides the conclusions of this paper.

2 PHYSICAL MODEL

2.1 Overview physical model

For this research a dedicated physical model, repre sented in 2.2, was built in test tank B of Flanders Hydraulics Research.

The physical model represents a part of a lock cham ber and a lock gate. The width of the physical model is 2.0 m. The downstream lock gate is not considered in the physical model. The feeding of the model is performed by a pumping system into the upstream reservoir. A lock gate with a thickness of 0.12 m is present between the upstream and the downstream reservoir (i.e. the lock chamber). Downstream of the lock gate, a measurement section with a length of 4.0 m is present. Both side walls of this measurement section consist of glass walls, allowing visualization of the flow pattern downstream of the opening in the lock gate using dye injection.

Above the measurement section a semi-automatic measurement robot is built, allowing to perform auto matic point velocity measurements. Downstream of the measurement section a honeycomb structure is positioned, in order to smooth the water level in the downstream reservoir. At the downstream end of the model a V-notch gate is used to implement a constant downstream water level. The V-notch gate is also used as a complementary discharge measurement. The physical model is meant to be a "generic" physical model and not a reproduction of an existing navigation lock. Therefore the design of the physical model was based on several design projects of Flemish inland navigation locks and the available space in the test tank. From this a diameter of the opening of 0.20 m was derived. However, some components of the physical model, namely the breaking logs, the gate thickness (0.12 m) and the height of the opening above the bottom (0.18 m) are based on the detailed design of the filling and emptying system for the new lock of Sint-Baafs-Vijve (Belgium). Scaling the diameter of the opening in the scale model to the openings of the design of the new lock of Sint-Baafs-Vijve, i.e 1.35 m, yields a scale factor 1:6.75. During the physical model tests, the water level of the upstream and the downstream reservoir was measured using two micropulse trandrucers (from Baluff). The discharge feeding the upstream reservoir was also measured using an electromagnetic current meter (from Krohne). Flow velocity measurements (see section 5) were carried out both with an electromagnetic velocity meter (EMS, from Deltares) and with an acoustic doppler velocimeter (Vectrino-II profiler, from Nortek). For the visualisation of the flow pattern in the lock chamber, dye (Potassium Permanganate) was injected upstream of the opening in the lock gate. A system equipped with 6 cameras (IDS UI-5490 RE and IDS UI-5420 RE), a power supply, data storage and 4 computers for data acquisition, was used to obtain sequential photos of the dye spreading. To provide a proper illumination of the measurement section 14 halogen spotlights with 90 W each were used, from which 12 spotligths were placed behind the left glass wall and 2 were mounted above the water surface. The local reference system used in this paper is also indicated in white in Figure 2. The jet exit is situated at $x = 0$, the symmetry plane of the physical model is situated at $y = 0$ and the centre of the opening in the gate is situated at $z = 0$. Consequently the bottom of the lock chamber is situated at $z = -0.18$ m. 2.2 Geometry opening in lock gate and studied breaking log configurations The gate opening, built in acrylic glass, contains a rectangular section, in which the cylindrical tube is positioned (Figure 3). The diameter of the circular opening is 0.200 m. The width of the downstream rectangular section is 0.278 m and the height is 0.286 m. Different configurations were tested in the physical model, varying the following parameters: presence of the breaking logs, position of the breaking logs (inside the lock gate, shown in Figure 3, or outside the lock gate), the number of breaking logs and the distance between the breaking logs. All tests with breaking logs described in this paper concern breaking logs with a square section.

Figure 2. Physical model; The local reference system used in this paper is indicated in white.

Figure 3. Gate opening and breaking logs (physical model

dimensions).

The physical model tests in this paper concern the

6 following configurations of breaking logs:

• NO: configuration without breaking logs

• IN: 3 breaking logs (width: 0.022 m) positioned

inside the lock gate. Distance between breaking

logs: 0.0450 m.

• OUT1: 3 breaking logs (width: 0.022 m) positioned

outside the lock gate. Distance between breaking

logs: 0.0450 m.

• OUT2: 3 breaking logs (width: 0.022 m) positioned

outside the lock gate. Distance between breaking

logs: 0.0249 m. • OUT3: 4 breaking logs (width: 0.022 m) positioned outside the lock gate. Distance between breaking logs: 0.0224 m. • OUT4: 5 breaking logs (width: 0.022 m) positioned outside the lock gate. Distance between breaking logs: 0.0150 m. Concerning these configurations the following is noted: • The configurations NO (without breaking logs), IN and OUT1 (with 3 breaking logs) were considered as base configurations during the research. Configurations OUT2, OUT3 and OUT4 are alternative configurations to study the influence of the number of breaking logs and the distance between the breaking logs. • The distance between the 3 breaking logs for configuration IN and OUT1 meets the value of 0.30 m (in prototype dimensions, using the aforementioned scale factor for the lock of Sint-Baafs-Vijve) recommended in Dutch literature (Beem et al. 2000). The recommended blockage of the opening of 50% is not met for these configurations. • For configuration OUT2, the distance between the breaking logs was reduced in order to divide the circular opening in 4 areas with an approximately equal surface. • The breaking logs for configuration OUT3 and OUT4 are equally spaced along the diameter of the circular opening in the lock gate. For these configurations the blockage of the (circular) opening is almost equal, respectively higher than 50 %, being the value recommended in Dutch literature (Beem et al. 2000). Table 1 summarises the characteristics of these 6 breaking logs configurations.

Table 1. Characteristics of tested breaking log configurations (N/A = not applicable).

Configuration name		NO	IN	OUT1	OUT2	OUT3	OUT4
Breaking logs		NO	YES	YES	YES	YES	YES
Number breaking logs	–	N/A	3	3	3	4	5
Width breaking logs	m	N/A	0.022	0.022	0.022	0.022	0.022
Distance between breaking logs	m	N/A	0.045	0.0450	0.0249	0.0224	0.0150
Position breaking logs in lock gate		N/A	Inside	Outside	Out-side	Outside	Outside
Blockage down-stream circular opening	%	N/A	35	35	39	48	59
Blockage upstream square opening	%	N/A	24	24	24	32	40

3 INFLUENCE OF BREAKING LOGS ON DISCHARGE COEFFICIENT OF GATE OPENING

3.1 Method

The (dimensionless) discharge coefficient μ is an important factor in numerical simulations of the filling discharge through gate openings during the design of a filling and emptying system. The discharge through a gate opening is described by the following formula:

where Q = discharge [m^3/s]; μ= discharge coefficient [–];A = is the cross-sectional area of the opening under the valve [m^2]; g = the gravitational acceleration [m/s^2] and Δh = the head [m].

At a certain time step the cross-sectional area A of the opening under the valve is calculated as follows

from the lift height of the valve:

where A = is the cross-sectional area of the opening

under the valve [m^2]; δ = circumferential angle of the

opening [rad]; D = diameter of the circular opening

(=0.20 m) [m]; h_V = lift height of the valve [m].

In this paper the ratio between the cross-sectional

area under the valve A and the maximum area of

the circular opening A_{max} ($=\pi D^2 /$ = 0.0314 m^2) will

be further referred to as the relative opening A/A_{max}

(in %) of the lift valve Note that formula (1) for the

cross-sectional area under the valve will be used for

all types of lock filling configurations, i.e. even in

case breaking logs are used (at the downstream side of

the gate opening). This implies that the cross-sectional

area blocked by the breaking logs at the downstream

side of the gate will not be accounted for in the

calculation of the discharge coefficient

To assess the discharge coefficient, steady-state

tests were performed in the physical model with dif

ferent (constant) lift heights of the valve. During these

tests the discharge, as well as the upstream and down

stream water levels were kept as steady as possible. Figure
4. Variation of the discharge coefficient as a function of
the relative opening of lift valve for breaking log
configuration NO, IN and OUT1. These tests were all
performed with a downstream water level of 0.70 m and
applying both a head of 0.20 m and a head of 0.50 m. All

tests were performed twice. The discharge, upstream water level, downstream water level and lift height of the valve were recorded during a period of 90 s. For each test, the value of the discharge coefficient is computed as a time-average of the values computed for the individual time-steps. 3.2 Results For the three base configurations NO (without breaking logs), IN (3 breaking logs inside the lock gate) and OUT1 (3 breaking logs outside the lock gate) steady state physical model tests were carried out for 8 values of the lift height of the valve. Figure 4 presents the variation of the computed discharge coefficient for all these tests as a function of the relative opening of the lift valve. During the physical model tests, the alternative configurations of breaking logs OUT2, OUT3 and OUT4 were only tested for two relative lift heights of the valve (50% and 100%). Table 2 presents for a fully opened lift valve the discharge coefficients of these alternative configurations together with the values for the standard configurations (NO, IN and OUT1). Table 2. Computed discharge coefficients for relative lift height 100%. NO IN OUT1 OUT2 OUT3 OUT4 Mean 0.66 0.68 0.70 0.66 0.68 0.63 Standard deviation 0.005 0.004 0.004 0.004 0.005 0.006

To quantify the variability of the computed discharge coefficients around the time-averaged value (of the 90 s time series), Table 2 mentions also the value of the standard deviation of the discharge coefficient for the different tests. The values of the standard devia tion show a low variability of the discharge coefficient around the computed average value.

For a relative opening of the lift valve of 30% and 40% Figure 4 shows a low variation of the discharge coefficient between the three studied configurations. The variation is higher for the low relative opening of the lift valve of 10% and 20%. At this lower relative openings, the measurement accuracy of the discharge and the measurement accuracy of the relative opening in the gate is lower. This is a possible explanation for the higher variation in discharge coefficients for these low relative openings of the lift valve.

For relative openings of the lift valve superior to 50% some (initially) unexpected results are noticed in Figure 4 and also (for the case of a fully opened valve) inTable 2. Introducing 3 breaking logs outside the lock gate (configuration OUT1) results into an increase of the discharge coefficient compared to the configura tion without breaking logs (NO).The inside position of the breaking logs (configuration IN) yields a slightly lower value of the discharge coefficient (μ= 0.68 for fully opened valves) than the outside position (OUT1; μ= 0.70 for fully opened valves), but still a slightly higher value than the configuration without breaking logs (NO; μ= 0.66 for fully opened valves).

The discharge coefficients computed from the phys ical model experiments are first compared with the discharge coefficient computed from water level mea surements, carried out in 2011 by Flanders Hydraulics Research in the inland navigation lock of Evergem near Ghent (Belgium). The filling and emptying sys tem of the latter lock (width: 25 m; length: 230 m; head: 1.02 m) consists of 12 circular openings in the mitre gate, similar to the configuration studied in this research.The openings are sealed with lift valves and 3 breaking logs are present inside the lock gate. From the water level measurements in the lock chamber during filling of the lock an average (over various lock fill ings) discharge coefficient of μ= 0.72 was computed for fully opened lift valves with a standard devia tion of 0.05 (computed for the period the valves are fully opened and for the various lock fillings). The discharge coefficient from the measurements in the lock of Evergem is slightly higher than the value of μ= 0.68 measured in the present physical model (for the configuration IN).

Secondly, the discharge coefficients from the phys

ical model tests are compared with the more general recommendations mentioned in the manual of the software Lockfill (Deltares 2015). This manual recommends the use of values between μ= 0.60 and μ= 0.90 for the discharge coefficients of openings in a lock gate sealed with vertical lift valves. The discharge coefficients from the physical model tests are situated within this range. The discharge coefficients for the configurations IN (3 breaking logs inside the lock gate) and OUT1 (3 breaking logs outside the lock gate) being higher than the value for the configuration NO without breaking logs, is in contrast with the results of Van der Ven et al. (2015). Van der Ven et al. (2015) shows lower discharge coefficients for configurations with breaking logs. Note that the geometrical configuration, the number of breaking logs and the distance between the breaking logs studied in the latter research – a rectangular opening sealed by a lift valve, widened towards the downstream end of the gate and 4 breaking logs – is different from the one considered in this research. This comparison shows that the geometry of the opening in the lock gate influences the effect of breaking logs on the discharge coefficient. Table 2 shows also that the discharge coefficient computed for configuration OUT2 (μ= 0.66 for fully opened valves), with a reduced distance between the 3 breaking logs, is considerably lower than the value for configuration OUT1 (μ= 0.70 for fully opened valves), although the blockage of the circular opening is only slightly higher than the blockage for configuration OUT1. The value of the discharge coefficient for configuration OUT2 is even equal to the value for the configuration without breaking logs (NO). From these observations follows the preliminary conclusion that (when keeping the number of breaking logs constant) the discharge coefficient is influenced by the distance between the breaking logs. Also, depending on the distance between the breaking logs, it seems possible to define a configuration of breaking logs that results into a discharge coefficient of the opening that does not differ from the configuration without breaking logs. When the number of breaking logs (positioned outside the gate) is increased from 3 (configuration OUT1) to 4 (configuration OUT3) and 5 (configuration OUT4), the discharge coefficient for fully opened valves reduces from μ= 0.70 (OUT1) to a value of μ= 0.68 (OUT3) and μ= 0.63 (OUT4), respectively. This reduction is explained by the increased blockage of the opening by the breaking logs. The value of the discharge coefficient for configuration OUT3 (4 breaking logs) is still higher than the value for the configuration without breaking logs (NO) and also higher than the value for configuration

Figure 5. Visualisation of flow pattern in lock gate downstream of the opening for configuration NO, OUT1 and IN.

Downstream water level: 0.70 m – head: 0.20 m – fully opened lift gate.

OUT2 with 3 breaking logs, although the blockage for

configuration OUT3 is higher than the blockage for

configuration OUT2.

These observations allow to conclude that adding

breaking logs in the opening, hence increasing the

blockage of the opening in the gate by the breaking

logs, can lead to a reduction of the discharge coeffi

cient. On the other hand, reducing the distance between the breaking logs (while keeping the number of breaking logs constant) can also result into reasonably lower discharge coefficients, while the blockage (of the circular opening) only slightly increases. As already mentioned above, the conclusions drawn in this paragraph are strictly valid for the studied geometrical configuration of a circular tube in a rectangular chamber and for the considered square shapes of breaking logs. Other geometries of the opening in the lock gate might result in different influences of breaking logs on the discharge coefficient of the opening in the gate.

4 VISUALISATION OF FLOW PATTERN DOWNSTREAM OF GATE OPENING

For analysing the flow pattern downstream of the lock gate, figures of the dyed flow pattern captured with the camera the closest to the gate opening are compared. Figure 5 shows the changing of the flow pattern with the introduction of the breaking logs, both for the flow in a horizontal plane and for the flow in a vertical plane. All the pictures of this figure depict tests with a water height of approximately 0.70 m downstream of the gate opening, with a head of 0.20 m and a fully opened lift gate.

The effect of the breaking logs is clearly noticed from this figure. In the case of breaking logs out side the lock gate (OUT1), a part of the filling jet

is deflected laterally, i.e. in a direction perpendicu

lar to the centreline of the jet (panel 5 of Figure 5). It should be noted that the flow velocity of this laterally deflected jet is much lower than the flow velocity of the main jet from the opening in the lock gate. With breaking logs inside the gate opening (configuration IN), no lateral deflection is noticeable, but the jet becomes clearly wider as compared to the case without breaking logs (configuration NO; panel 6 of Figure 5). Panel 1, 2 and 3 of Figure 5 show clearly the widening of the jet in the vertical plane for both configurations with breaking logs (IN and OUT1), compared to the configuration without breaking logs (NO). Between both configurations with breaking logs (IN and OUT1) no significant difference in widening of the jet in the vertical plane is noticed. These observations suggest that the breaking logs (in a vertical position) enhance the spreading of the jet, which is considered to be beneficial for lock filling purposes. It should also be noted that in this part of the project only one opening in the lock gate is considered. However, in real locks multiple openings in the lock gate are usually present, constraining each other's flow pattern. The visualisation of the flow pattern in the lock chamber also shows a slight asymmetrical behaviour of the jet towards the downstream end of the physical model. This asymmetrical behaviour is visible in panel 5 of Figure 5. The asymmetric jet behaviour is also confirmed by the velocity measurements with the electromagnetic velocity meter (see section 5.2). 5 MEASUREMENT OF FLOW VELOCITY DOWNSTREAM OF THE GATE OPENING 5.1 Method To characterise the flow pattern downstream of the gate opening, point measurements of the flow velocity were carried out. Both an electromagnetic velocity

Table 3. Characteristic parameters of the electromag

netic velocity meter and the Vectrino-II profiler (N/A = not

applicable).

	Electromagnetic velocity meter	Vectrino II-profiler
Manufacturer	Deltares	Nortek
Velocity components	v_x , v_y	v_x , v_y, v_z
Horizontal velocity range	~2.5 m/s	~3.0 m/s

Vertical velocity range	N/A	1.0 m/s
Number of cells	N/A	9
Cell size	N/A	4 mm
Measurement frequency	50 Hz	50 Hz
Sampling rate	10 Hz	50 Hz
Measurement duration	240 s	180 s

meter (EMS, from Deltares) and an acoustic Doppler velocimeter (Vectrino II-profiler, from Nortek) were used to measure the flow velocity. The electromag netic velocity meter measures two components of the horizontal velocity (v_x and v_y) in a single point below the probe, for velocities up to 2.5 m/s. No vertical component of the velocity can be measured. The Vec trino II-profiler, measures the three components of the velocity (v_x ,v_y and v_z), for velocities up to approxi mately 3.0 m/s, in a profile with a height of 36 mm. Note that in the current research only the flow veloci ties in one particular cell of the measured profile (i.e. the sweet spot) are considered.TheVectrino II-profiler is consequently used for performing point measure ments rather than for measuring a velocity profile. Table 3 summarises the characteristic parameters of both instruments for measuring the flow velocity.

For the base configurations NO (without breaking logs) and IN (3 breaking logs inside the gate) the flow velocity was measured in a horizontal plane in the cen treline of the (circular) opening for the situation with a fully opened lift valve. These measurements were per formed using theVectrino II-profiler and are described in section 5.3.

As a first (and easier) comparison of the base con figurations (NO, IN and OUT1) with the alternative configurations (OUT2, OUT3 and OUT4), velocity measurements were carried out in the centreline of the opening at three distances from the gate using the electromagnetic velocity meter. The results of the flow velocity measurements for the situation with a fully opened lift valve using the electromagnetic velocity meter are described in section 5.2.

5.2 Measurement of flow velocity in three locations downstream of the gate opening

The flow velocity was measured in the centreline of the opening ($y = 0.0$ m, $z = 0.00$ m), at three distances ($x = 0.5$ m, $x = 1.0$ m and $x = 1.5$ m) downstream of the gate opening. A downstream water depth of 0.70 m was considered and a head of 0.20 m. The mea

surements were carried out with the electromagnetic velocity meter (EMS) during a period of 240 s using a sampling rate of 10 Hz. The data of the EMS were measured with a frequency of 50 Hz and have passed a second order filter at 10 Hz before being sampled (WL|Delft Hydraulics 2001). For the post-processing of the velocity measurements a moving average with a period of 1 s was applied to the measured flow velocities, in order to smoothen the data.

Afterwards the mean velocity over the measuring period of 240 s was computed. For the 6 tested configurations of breaking logs, the mean values of the xand y-component of the measured velocity at the three distances from the gate are presented in Figure 6. For analysis purposes and to facilitate a comparison with the behaviour of (3D or wall) jets in literature, the results are presented in a dimensionless form. The diameter of the opening D is used as length scale and the cross-sectionally averaged velocity U o under the (fully opened) lifting gate is used as velocity scale. The latter velocity is simply calculated as: where: U 0 = cross-sectionally averaged velocity under fully opened lift gate [m/s]; μ= discharge coefficient (Table 2) [-]; g = the gravitational acceleration [m/s 2] and Δh = the head (=0.20 m) [m]. At x/D = 2.5, Figure 6 shows for all configurations with breaking logs considerably lower values of the measured x-component of the flow velocity, indicating the jet spreading (as noticed in section 4). It should be noted that all the tested configurations with breaking logs contain a breaking log in the vertical centreplane of the opening in the gate. Hence the velocity measurements along the centreline of the opening might be in the wake of the central breaking log. This should be taken into account when interpreting the velocity measurements along the centreline. The highest flow velocity was measured for the configuration without breaking logs (NO). In the vicinity of the lock gate (x/D = 2.5) a velocity of 1.5 U 0 parallel to the axis of the physical model was measured. The flow velocity decreases with increasing distance to the lock gate until 0.36 U 0 at x/D = 7.5. The other base configurations IN (breaking logs inside the lock gate) and OUT1 (breaking logs outside the gate) show a decrease of the flow velocity in the x-direction, compared with the configuration NO without breaking logs. However, the measured velocity for both these configurations differ only slightly from each other. The alternative configuration OUT2, i.e. 3 breaking logs positioned outside the lock gate with a smaller distance between the breaking logs, shows a slightly lower velocity than the configuration without breaking logs, but a higher velocity than both the other configurations with 3 breaking logs (IN and OUT1). Similarly as for the discharge coefficient, the influence of configuration OUT2 on the (relative) flow velocity is less than for the configurations IN and OUT1, although the blockage of the circular opening is slightly higher.

Figure 6. Comparison of mean values of xand ycomponent of flow velocity (v x and v y) measured with the EMS in the

centreline of the opening ($y = 0.0$ m; $z = 0.0$ m). Increasing the number of breaking logs to 4 (configuration OUT3) and 5 (configuration OUT4) results into a further decrease of the velocity downstream of the lock gate, because of the increased blockage of the opening. The configuration with 4 breaking logs (OUT3) results into lower velocities than the configuration with breaking logs inside (IN), although the discharge coefficient for these both configurations is the same. For the configuration with 5 breaking logs (OUT4) velocities even lower than $0.01\ U_0$ are measured in the vicinity of the lock gate.

For the configurations with breaking logs, in general, lower flow velocities are measured downstream of the lock gate, indicating the spreading of the jet. Increasing the blockage of the (circular) opening by increasing the number of breaking logs results into a decrease of the flow velocity downstream of the opening. However, reducing the distance between the breaking logs (and keeping the number of breaking logs constant) can even result in an increase of the flow velocity, although the blockage of the circular opening is slightly higher.

Figure 7 presents the time variation of the y component of the flow velocity v_y at the three con sidered measurement locations ($x/D = 2.5$; $x/D = 5.0$ and $x/D = 7.5$) for the configuration without breaking logs (NO). Due to the measurement method – only one measurement device is used for performing the velocity measurements – Figure 7 presents 3 (in time) separate velocity measurements. This should be taken into account when comparing the time variation of the measurements for one location to the one for another location.

Figure 7 shows a slight constant negative velocity v_y at $x/D = 2.5$. The time variation of this velocity

Figure 7. Time variation of flow velocity v_y in the centreline of the opening ($y = 0.0$ m; $z = 0.0$ m). Note: measurement at each position is a different test (in time)

shows a small cyclic behaviour with an amplitude of approximately $0.1\, U_0$ and a period of approximately 60 s.At the more downstream locations ($x/D = 5.0$ and $x/D = 7.5$) this cyclic behaviour is more clearly visible. The amplitude increases also with increasing distance from the gate opening to $0.2\, U_0$ at $x/D = 5.0$ and $0.4\, U_0$ at $x/D = 7.5$. The period of this cyclic behaviour is almost the same for the both locations. This cyclic and asymmetric behaviour of the jet (also noticed in section 4) is possibly due to the backflow, produced due to boundary conditions of the lock chamber, like e.g. the honeycomb structure (see Figure 2) downstream. Narrowing the lock chamber in the model is most likely one option to reduce the jet asymmetries, hence simplifying the analysis and coming closer to a realistic jet development in lock chamber filling (where the jets exiting from multiple, adjacent openings constrain each other’s trajectories).

Figure 8. Flow velocity (in m/s) in x-direction (v_x, top panels) and y-direction (v_y lower panels), measured with the Vectrino

II-profiler in a horizontal plane in the centreline of the opening ($z = 0.00$ m).

5.3 Velocity measurements in a horizontal plane in the centreline of the opening

The velocity measurements in a horizontal plane in the centreline of the opening (located at z = 0.00 m) were carried out using the Vectrino II-profiler with a down stream water level of 0.70 m and a head of 0.50 m. The velocity measurements were only carried out for configuration NO (without breaking logs) and config uration IN (breaking logs inside the gate) for a fully opened lift valve in a (rather) coarse grid of points. Because the maximum measured velocity for these configurations is close to the maximum velocity range of the Vectrino II-Profiler, a phenomenon known as velocity aliasing occurs. The measured data were first de-aliased with a script based on the script by Gees (2013), based upon the algorithm by Parkhurst et al (2011). Afterwards the data were despiked using a procedure similar to the one proposed by Goring and Nikora (2002) and discussed by Wahl (Wahl 2003). Figure 8 presents for both configurations (NO and IN) a vector map of the velocity components in the x and y direction. Every circle represents, by grayscale, the velocity either in xor y-direction. The vector rep resents the direction and magnitude of the velocity in the horizontal plane considered. In the different panels of the figure the average velocity over the measurement period of 180 s is represented.

Figure 8 shows clearly the decay of the velocity along the jet centreline. Near the filling opening, some entrainment can be noticed, i.e. y-components pointing towards the jet centreline. This is especially visible in the left panels of Figure 8 for configuration NO without breaking logs. More downstream, jet spreading is Figure 9. Velocity decay along the centreline of the opening. noticeable, i.e. y-components pointing away from the jet centreline. This is especially visible in the left panels of Figure 8 for configuration IN with 3 breaking logs inside the gate. Figure 9 compares the decay of the velocity along the jet centreline from the measurements with the Vectrino II-profiler for configuration NO without breaking logs with the velocity decay for configuration IN with 3 breaking logs inside the gate. For analysis purposes the results are presented in the same dimensionless form as in section 4.2, using the diameter of the circular opening as length scale and the velocity U_0 (computed for a head of 0.50 m) as velocity scale. For the configuration without breaking logs, the flow velocities from section 5.2 measured with the electromagnetic velocity meter in the three locations on the centreline of the jet are also presented in Figure 9 using the gray coloured symbols.

The comparison of the flow velocities measured with the Vectrino II-profiler and those measured with the electromagnetic current device shows that both instruments measure similar velocities. This observation provides confidence in the velocity de-aliasing that was carried out on the data of the Vectrino II-profiler.

For the configuration NO without breaking logs Figure 9 shows a velocity of 1.5 U_0 immediately downstream of the lock gate. The reason for this velocity being higher than the cross-sectionally averaged veloc ity under the lift gate is twofold. First, contraction of the flow at the edge of the circular opening and sec ond, entrainment of water towards the jet is resulting into an increase of the jet flow velocity. Further down stream, the flow velocity decreases towards 1.0 U_0 near $x/D = 3.0$ and 0.5 U_0 near $x/D = 5.0$.

When breaking logs are present (inside the gate; configuration IN), the flow velocity in the x-direction immediately downstream of the gate is almost half of the flow velocity for the case without breaking logs (NO). This lower velocity is explained by the spread ing of the jet because of the presence of the breaking logs (as already noticed in section 4). For measurement locations in the vicinity of the lock gate, similarly as for the electromagnetic current devise (see section 5.2), the Vectrino II-profiler might be situated in the wake of the (middle) breaking log, inducing a lower velocity. This should be taken into account when interpreting the velocity measurements.

The decay of the flow velocity for configuration IN (with breaking logs) is more linear shaped as compared to the more exponential decay of the configuration NO (without breaking logs). More downstream than $x/D = 7.0$ even higher velocities were measured with

breaking logs (IN) than without breaking logs (NO).
The presence of the honeycomb structure near the end of the physical model (see Figure 2) and back flow effects (see section 4 and section 5.2) are a possible explanation for this higher velocity near the downstream end of the physical model.
Figure 9 shows also a region with an almost con stant velocity immediately downstream of the opening, indicating the core of the jet. The length of this region is approximately equal to 1.0 D for configuration NO without breaking logs and 2.0 D for configuration IN with breaking logs inside the gate. These values are considerably lower than the values between 3.5 D and 6.0 D found in literature (Rajaratnam 1976, Grand champ et al. 2013, Nyantekyi-Kwakye et al. 2015).
It should be noted that in literature typically a virtual origin, not coinciding with the jet exit (mostly from nozzles), is considered to define the length of the core of the jet. In the present research the length of the core of the jet is measured from the downstream end of the lock gate. This difference hampers the direct compari son of the results of the present research with literature results.
In general, an exponential decay of the velocity along the centreline for the configuration without

breaking logs is observed. The configuration with breaking logs inside the lock gate shows lower velocities in the vicinity of the lock gate, indicating the jet spreading (also noticed in section 4), and shows a more linear decay of the velocity along the centreline. The length of the core of the jet is found to be significantly lower than the values reported in literature, although the comparison is hampered by differences in the definition of the (virtual) origin of the jet. 6 CONCLUSION To investigate the influence of breaking logs on the flow pattern and energy dissipation in the lock chamber a dedicated (generic) physical model was built, consisting of one circular opening in a lock gate sealed by a vertical lift valve. Using the physical model, 6 configurations were tested, differing from each other in the presence of (square) breaking logs, the distance between the breaking logs and the number of breaking logs. The influence of the breaking logs on the discharge coefficient of the opening and on the flow velocity in the lock chamber was assessed. The discharge coefficients derived from the physical model experiments are comparable to the ones derived from water level measurements in the inland navigation lock of Evergem (near Ghent, Belgium) and to the values mentioned in literature. Concerning the discharge coefficient and the flow velocity downstream of the lock gate, the physical model experiments show both logical (e.g. lower velocity in the lock chamber when introducing breaking logs) and counter intuitive results (e.g. discharge coefficients with breaking logs being higher than without breaking logs). The velocity measurements and the visualisation of the flow pattern show at short distances from the gate opening the spreading of the jet and the velocity decay along the centreline. The length of the core of the jet was found to be significantly lower than the values reported in literature, although the comparison is hampered by differences in the definition of the (virtual) origin of the jet. Also an asymmetrical behaviour of the jet and backflow effects were noticed, due to the dimensions and downstream boundary conditions of the model. The physical model experiments show some first observations of the influence of breaking logs on the flow pattern in the lock chamber and the jet behaviour valid for the studied geometrical configuration of a circular tube in a rectangular chamber and for the considered square shapes of breaking logs. Nevertheless, a more intensive (i.e. on a finer grid) measurement of the flow velocity in the lock chamber and a visualisation of the flow pattern in the lock gate itself will be carried out, in order to acquire a deeper understanding of the behaviour of the jet from an opening in the lock gate when breaking logs are present. Afterwards also alternative configurations of breaking logs (e.g. with different cross-sectional shapes) or an

alternative geometry of the opening in the lock gate

will be investigated in the physical model.

Beem, R.C.A., Boogaard, A., Glerum, A., de Graaf, M.A.,

Henneberque, S.D., Hiddinga, P.H., Kranenburg, D., van

der Meer, M.T.J., Nagtegaal, G., Van der Paverd, M.,

Smink, L.M.C., Vrijburcht, A., and Weijers, J., 2000.

Design of locks. Utrecht: Ministery of Transport, Pub

lic Works and Water Management. Civil Engineering

Division.

Deltares, 2015. LOCKFILL. User and Technical manual.

version 0.00 8 July 2015.

Gees, J., 2013. In Stream Vegetation Interactions with Three

Dimensional Flow Fields. Undergraduate Dissertation.

University of Hull (UK).

Goring, D.G. and Nikora, V.I., 2002. Despiking Acoustic

Doppler Velocimeter Data. Journal of Hydraulic Engi

neering, 128, 117–126.

Grandchamp, X., Van Hirtum, A., and Pelorson, X., 2013.

Centreline velocity decay characterisation in low-velocity

jets downstream from an extended conical diffuser.

Meccanica, 48, 567–583. Nyantekyi-Kwakye, B., Tachie, M.F., Clark, S.P., Malenchak, J., and Muluye, G.Y., 2015. Experimental study of the flow structures of 3D turbulent offset jets. Journal of Hydraulic Research, 53 (January), 773–786. Parkhurst, J.M., Price, G.J., Sharrock, P.J., and Moore, C.J., 2011. Phase unwrapping algorithms for use in a true real-time optical body sensor system for use during radiotherapy. Applied optics, 50 (35), 6430–6439. Rajaratnam, N., 1976. Turbulent jets. Amsterdam: Elsevier Scientific. Van der Ven, P.P.D., Van Velzen, G., O'Mahoney, T.S.D.., and De Loor, A., 2015. Comparison of scale model measurements and 3D CFD simulations of loss coefficients and flow patterns for lock levelling systems. In: SMART RIVERS 2015, Buenos Aires, Argentina, 7–11 September 2015: papers booklet. Wahl, T.L., 2003. Discussion of "Despiking Acoustic Doppler Velocimeter Data" by Derek G. Goring and Vladimir I. Nikora. Journal of Hydraulic Engineering, 129 (6), 484–487. WL|Delft Hydraulics, 2001. P-EMS. Programmable electronic liquid velocity meter. version 2.01. User-Manual. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Relation between free surface profiles and pressure profiles with respective fluctuations in hydraulic jumps

J.D. Nóbrega & H.E. Schulz

Department of Hydraulics and Sanitary Engineering, School of Engineering at São Carlos,

University of São Paulo, Brazil

M.G. Marques

Institute of Hydraulic Research, Federal University of Rio Grande do Sul, Brazil

ABSTRACT: Because the hydraulic jump is a dissipative singularity, it is used to dissipate energy for example

in stilling basins downstream of spillways. On the other hand, concerns about stilling basins are the possibility of

cavitation and uplift of baffle blocks, which are mainly related to pressure fluctuations, and the water depths along

the hydraulic jump, relevant for the design of the walls of the stilling basins. Interestingly, pressure fluctuations

and depth fluctuations are caused by the same turbulence in the roller. So, both phenomena may present similar

behaviors for the statistical properties. This paper analyzes instantaneous water depth data, measured using ultrasonic sensors, for inflow Froude numbers from 2.8 to 5.3. Statistical parameters of the water depth data were obtained and compared to data of pressure fluctuations found in the literature. As expected, it is shown that there are similarities between the behavior of both phenomena.

1 INTRODUCTION

Hydraulic jump (HJ) is the open flow phenomenon related to the transition of a supercritical to a sub critical flow condition. The hydraulic jump may be a useful phenomenon, for example in stilling basins downstream of dam spillways, where the presence of baffle and chute blocks imposes the hydraulic jump formation and intense local energy dissipation.

The hydraulic jump is usually described as a com plex phenomenon, being characterized by a rapid expansion of the free surface level. At the region of rapid expansion, a high amount of air enters into the flow in the form of bubbles. The air bubbles are then advected and diffused in the flow, and finally released back to the atmosphere, as the water flows downstream (Chanson, 2011). Another determinant characteris tic of hydraulic jumps is the presence of turbulence, causing intense fluctuations of flow parameters like velocity, surface position and pressure. Regarding the longitudinal position of the hydraulic jump, the front

of the jump moves back and forward in a very chaotic

movement. Similarly, the flow in the vertical direc

tion changes rapidly in time, producing many splashes

and droplets, which reach heights that may exceed the

value of the vertical dimension of the jump itself.

The water surface mean profile (evolution of the

mean water depth with the longitudinal distance)

may be viewed as being composed firstly by a “sat

uration exponential” profile which then rises until

the condition of uniform flow (L_j – hydraulic jump

length). In the first region, the flow usually presents a
Figure 1. Hydraulic jump. * L_r = roller length; L_{tr} = transition length; and L_j = hydraulic jump length.
recirculation pattern (L_r – roller length), with the water at the vicinity of the free surface tending to move down in the opposite direction of the main flow. Beneath the roller region, the main flow stream expands in the downstream direction (see Fig. 1). Due to the complexity of the hydraulic jump phenomenon, many of its characteristics are still not completely understood, which explain the need of continuous studies in this theme. Some of the issues that have been more recently addressed are: 1. numerical simulation of the flow (Chern & Syamsuri 2013, Simões et al. 2010) 2. influence of the bed roughness on flow patterns (Carollo et al. 2007, Ead & Rajaratnam 2002) 3. performance of baffle blocks (Habibzadeh et al. 2012, 2014) 4. air-water interface characteristics (Murzyn et al. 2007, Nóbrega et al. 2014, Wang et al. 2015) 5. properties of the airwater flow (two-phase flow) – (Chanson 2011, Zhang et al. 2014) 6. determination of the turbulent field using PIV (particle image velocimetry), BIV (bubble image velocimetry) and ADV (acoustic Doppler velocimeter) – (Lennon & Hill 2006, Lin et al. 2012,

Liu et al. 2004, Mignot & Cienfuegos 2011) 7. the

oretical and semi-empirical formulations (Beirami &

Chamani 2010, Schulz et al. 2015, Valiani 1997).

Regarding the pressure field generated by hydraulic jumps, an expressive number of researchers carried out experimental investigations, some of them devoted to describe the evolution of the pressure along the bot tom of the channel. The first papers on this theme were probably published by Vasiliev & Bukreyev (1967), Wisner (1967), and Abdul Khader & Elango (1974).

Abul & Elango (1974) used pressure cells on the bot tom of the channel for three series of experiments in order to measure mean pressures and their fluctuations in the center line of the jump.

Bowers & Toso (1988) also presented information about pressure fields in order to analyze the possi ble causes of failure of the Karnafulli spillway. Based on their experiments in a physical model, they dis cussed that the slab failure was probably caused by the differences in fluctuating pressure at the chute of the spillway and the chute slab. Additional information of maximum and minimum instantaneous pressure distri bution along the flow, in relation to mean pressure, was further presented byToso & Bowers (1988). Moreover, they evaluated the influence of different physical and flow conditions (inflow Froude number, developed and undeveloped incident flow, chute slopes, chute blocks, intermediate blocks and end sill) on the pressure field. Fiorotto & Rinaldo (1992) also developed studies on fluctuating pressure in hydraulic jumps in view of the stability of the stilling basin. The paper gives infor mation about statistical parameters along hydraulic jumps, covering a broader range of inflow Froude num bers than the previous works. Armenio et al. (2000) also evaluated statistical quantities of pressure fluctu ations at the bottom of a hydraulic jump over a negative step, furnishing results of extreme values and spa tial correlation structures. Studies on different aspects of pressure fluctuations and their decaying, pressure distribution, among other relevant information about the pressure fields were conducted by Marques and coworkers, like: Marques et al. (1997), Marques et al. (2000), Neto & Marques (2008).

In relation to the position of the free surface in hydraulic jumps, some authors have focused their attention on the investigation of the dynamics of the air-water interface, viewed as dependent of interac tions between the large-scale eddies and the free surface. Murzyn et al. (2007) worked on the theme and used wire gauges in their experimental studies. Besides determining the free surface and relevant tur bulence profiles, they furnished free surface length scales for both longitudinal and the transversal direc

tions. Kucukali & Chanson (2008), Chachereau &

Chanson (2011), and Murzyn & Chanson (2009) per

formed measurements using acoustic displacement

meters, described as a non-intrusive technique. Simul

taneously, two-phase flow properties were recorded

using phase detection probes. Their results include:

free surface profiles, free surface fluctuations, spectral

analysis of the data obtained with the phase detection probes and the acoustic sensors, integral turbulent time and length scales, and Strouhal numbers. The study of Nóbrega et al. (2014) showed a good comparison between mean free surface level from ultrasonic sensor measurements and images, which were obtained using a high speed camera focusing the flow from the sidewall. Because of the influence of the strong turbulence on both the deformation and breakup of the free surface and on the pressure fluctuations in hydraulic jumps, it is expected that the statistical quantities of the free surface fluctuations in hydraulic jumps may be correlated to those of the pressure fluctuations on the bottom of the flume. However, there is not a study that provides this comparison and quantitative analyses for this relationship. Therefore, this paper aims to furnish statistical information about the free surface position and the pressure field along hydraulic jumps, comparing the free surface measurements using ultrasonic sensor and the pressure data from the literature. The results may conduce to indirect evaluations of the behavior of mean statistical characteristics of the pressure evolution based on observations of the free surface. 2 EXPERIMENTAL SETUP 2.1 Facilities and flow conditions The experiments about the evolution of the surface were conducted in two recirculating channels: one at the Hydraulics Laboratory of the Engineering School of São Carlos – University of São Paulo, Brazil; and the other at the Undergraduate Laboratory of the University of Alberta, Canada. The channels are identified here as 1 and 2, respectively, as shown in Figure 2. Channel 1 is 41 cm wide, 60 cm height and has bed and sidewalls built in concrete. The flow rate was measured by a triangular weir, displaced downstream of the water tank. Hydraulic jumps were produced downstream of a broad crested weir of 24 cm height and 46 cm long. Channel 2 is 50 cm wide, 5 m long and has plexiglass side walls and an aluminum bed. The flow rate was measured by a magnetic flow transmitter (Foxboro ® Model IMT25) connected to the recirculation pipe. The supercritical condition was formed as the water passed under a sluice gate at the flume entrance, with an opening of 2.8 mm. The position of the hydraulic jump in both channels was controlled by a tail gate. The mean front of the hydraulic jump was positioned at the maximum water jet contraction downstream of the broad crested weir or sluice gate. Seven hydraulic jumps with inflow Froude numbers from 2.8 to 5.3 were tested. Table 1 presents the flow conditions. In all studies the possibility of scaling is always a present question. According to Murzyn & Chanson (2008), for hydraulic jumps with Re_1 up to 1×10^5, the rate of entrained air and air-water interfacial area

Figure 2. Experimental apparatus: a) Flume 1: broad crested weir; b) Flume 2: sluice gate.

Table 1. Experimental flow conditions.

Flume 1						
Broad crested Weir	Q	y_1	y_2	V_1		
Code	L/s	cm	cm	m/s	Re_1	F_1
II-21	21.0	2.5	14.2	2.04	51,000	4.11
II-31	31.0	3.5	17.4	2.15	75,250	3.67
II-40	40.0	5.0	21.4	1.95	97,500	2.78
Flume 2						
Sluice gate	Q	y_1	y_2	V_1		
Code	L/s	cm	cm	m/s	Re_1	F_1
III-21	20.9	2.8	10.6	1.56	43,680	2.98
III-27	26.9	2.9	14.2	1.94	56,260	3.64
III-34	34.2	2.9	17.9	2.44	70,760	4.58
III-39	38.9	2.9	20.5	2.80	81,200	5.26

* Q = flow rate; y_1 = supercritical depth; y_2 =

subcritical

depth; V_1 = supercritical velocity; Re_1 = supercritical

Reynolds number; F_1 = supercritical Froude number.

are underestimated. The authors also discuss that

the dynamic similarity of two-phase flow cannot be

achieved unless working in a full scale 1:1. Teixeira

et al. (2012), and Teixeira (2008) conducted pressure

measurements in spillways, for the Porto Colômbia

Dam in the prototype and in scales 1:32, 1:50, 1:80,

1:100. Their investigation showed that the mean pres

sure behaviors could be studied in the 1:100 scale

without loss of information. Because pressure and

depths are correlated, but also air entrainment may

affect this correlation, future investigations regarding

the scale effects are still welcomed.

2.2 Instrumentation

An ultrasonic sensor (Vernier ® – Go!Motion) was

used to detect the free surface position along the

hydraulic jumps. This sensor was also previously used
Figure 3. Ultrasonic sensor in a displacement track
(detail). by Simões et al. (2010, 2013), in their studies
on hydraulic jumps and stepped spillways. The ultrasonic
sensor was mounted in a displacement track at the central
longitudinal axis of the flume, above the free surface, in
order to determine instantaneous positions of the surface
(Fig. 3). The sampling time and frequency for the
measurements at each position were 2.0 minutes and 25.0 Hz,
respectively. The range of distance measured by the sensor
is from 15 to 600 cm, with a resolution of 1 mm. According
to the manufacturer, the emitted sound waves travel at a
speed of about 343 m/s, forming a conical frustum with
angle between 15 and 20 degrees and smaller base diameter

of 3.7 cm, corresponding to the senser/detector. The sampling was accomplished by a Logger Lite –Vernier ® software.The ultrasonic sensor records the time for the sound wave emitted by the sensor (in a conical frustum region) to return to the device, after reaching the air-water interface or its splashes. This travel time is proportional to the distance between the sensor and the obstacle, hence instantaneous positions of the obstacles are registered, being the water depths statistically determined in the sequence. Details about the statistical procedures followed to obtain the mean water depths at each measurement position may be found in Simões (2012) and Nóbrega (2014). 3 RESULTS 3.1 Free surface profile Although the free surface profile of hydraulic jumps are very unstable, both in horizontal and vertical directions, due to the jump toe oscillations and high level of turbulence, a mean profile can be defined from the time series for each location obtained by the ultrasonic sensor. The time series in the experiments were composed by 3000 data for each position. Figure 4 shows the mean depths along a hydraulic jump for $F_1 = 3.64$,

Figure 4. Instantaneous depths and mean profiles for

hydraulic jump with $F_1 = 3.64$.

the points representing the instantaneous values, and

blue curves representing the mean value ±3 times

the standard deviation. As can be observed in Figure

4, the standard deviation increases within the roller

length ($x - x_1 \approx 50$, defined by visual observations),

and gradually decreases afterwards. The maximum

depth ($y = 16.5$ at $x - x_1 \approx 60$), considering the mean

value plus 3 times the standard deviation, is about

16% higher than the subcritical depth $y_2 = 14.2$, for

$F_1 = 3.64$.

In order to define a representative curve for all

experimental data, the free surface profiles were plot ted in a nondimensonal graph. Figure 7a shows that for hydraulic jumps with $F_1 = 4.58$ and 5.26, the free surface profile presented a wavy shape. A possible cause may be the proximity of the jump toe to the sluice gate. For the other inflow Froude numbers tested here, the mean surface profiles of the hydraulic jumps are smooth and gradually rise until the uniform flow condition (and further decrease accordingly the flow conditions).

Some authors have proposed experimental solution for the free surface. Hager (1993), for example, defined an expression for the dimensionless depth as a function of the dimensionless length x/L_r. Other expressions for free surface profiles were proposed by Schulz et al. (2015), which were adjusted to the same experimental data presented here.

Visually comparing the mean depths to the mean pressures along hydraulic jumps (Figures 7a and 5), similar main patterns are observed. However, the curve of mean pressure attains $(P_x - y_2)/(y_2 - y_1) \approx 1$ near the position $x/(y_2 - y_1) \approx 8$, while the curve of mean depth attains the value of $(y - y_2)/(y_2 - y_1) \approx 1$ at $x/(y_2 - y_1) \approx 6$ without considering the two oscillating surfaces. If considering the oscillating cases, the value $x/(y_2 - y_1) \approx 8$ is also observed.

Figure 7b presents the mean and extreme depth val

ues for the tested flow conditions. It can be observed

that near the jump toe, some negative values of water

depths are recorded, which are unreal. Negative values

may be caused by different positions in the measure

ment cone region. Considering an angle of 20 degrees

for the cone, distances up to 1.5% larger than the ver

tical height may be registered for horizontal surfaces. Figure 5. Mean pressure distribution along hydraulic jump. Marques et al. (1997). Figure 6. Mean, maximum and minimum pressures along hydraulic jump. Marques et al. (1997). Additionally, multiple obstacles (drops) may generate reflections between them before attaining the detector, implying in larger travel times, and, consequently, larger calculated distances. Drops are caused by the strong oscillations of the jump in the longitudinal and vertical directions, so that the sound in the conical frustum may be subjected to the mentioned reflections. The maximum extreme values form a cloud of points which is more dense until about 1.5 times the dimensionless subcritical depth $(y - y_2)/(y_2 - y_1) = 1$, occurring in the range of positions $x/(y_2 - y_1) \approx 2$ to 6, approximately. The maximum differences between maximum extreme depths and mean depths were found at $x/(y_2 - y_1) \approx 1$. Comparing these behaviors and values with the results of Figure 6, which shows the maximum, minimum and mean dimensionless pressure, we note that downstream $x/(y_2 - y_1) \approx 8$, the extreme values of depth and pressure become approximately constant. The maximum pressure events concentrate around $x/(y_2 - y_1) \approx 2$. The behaviors of the events of minimum pressure and minimum depths, although of course located under the points representing the mean values of both variables, evolve differently concerning the length of the concavity of the subjacent curve. One reason of this difference was already mentioned, that is, depths have no negative values (negative measurements may be related to the conical frustum measurement region and to reflections of the sound in it). Another reason may be the fact that the pressure field is affected by the flow separation at the flume bottom.

Figure 7. a) Mean free surface profile; b) Minimum and maximum depths recorded using ultrasonic sensor; c)

Fluctuation

coefficient depth distribution; d) Skewness depth distribution along the hydraulic jump.

3.2 Fluctuation coefficient

The fluctuation coefficient relates the mean standard

deviation of the depth data set at a certain position – C'_y

(or pressure data set – C'_p) to the inflow kinetic energy

at the toe of the jump.

where σ_y = standard deviation of the depth data set;

σ_p = standard deviation of the pressure data set;

V_1 = inflow velocity; and g = gravity.

Towards the tail of the jump, the fluctuation coeffi

cient presented approximately a constant value, around

0.02 (Fig. 7c). In general, the maximum value of the

fluctuation coefficient is about 0.08, at the position

$(x - x_1)/(y_2 - y_1) \approx 1$. One exception was however

observed: experiment I 40, with $F_1 = 2.78$, presented

a different behavior, with the maximum value of 0.13.

These results are similar to those presented by Neto

& Marques (2008), and Lopardo (2012), for pressure

fluctuations (Fig. 8).The maximum fluctuation coeffi

cient of Neto & Marques (2008) experiments, based on

the mentioned pressure measurements (using pressure

transducers), were of about 0.08 at position $x/y_1 \approx 10$,

for hydraulic jumps with $F_1 = 4.55$ and 5.01. Accord ing to Lopardo (2012) measurements, the maximum values of C'_p were also around 0.06–0.07, at position $(x - x_1)/(y_2 - y_1) \approx 2$.

For both variables (depth and pressure), downstream the position $(x - x_1)/(y_2 - y_1) \approx 10$, the coefficients become approximately constant. The value obtained for depth measurements is about 0.02 (see Fig. 7c), which however is higher than the value of 0.01 found by Neto & Marques (2008), Lopardo (2012), and Abdul Khader & Elango (1974), for the decay of the standard deviation of the pressure along the jump (Fig. 8). This is an interesting conclusion, because it evidences that both measurements (depth and pressure) may be used to limit the length of the hydraulic jump. Because asymptotic behaviors are observed, conditional definitions of lengths may be used involving mean values or extreme values.

3.3 Skewness

The skewness is the third central moment and is a quantity that represents the asymmetry of the probability distribution of the measured instantaneous values in relation to the mean value.

where y_i = instantaneous depth at location x; y_x = mean value at location x; n = length of the sample data; σ_x = standard deviation at location x.

As can be observed from Figure 7d, for the depth

measurements, the skewness distribution is positive Figure 8. Fluctuation coefficient pressure distribution along hydraulic jump.Adapted from Neto & Marques (2008), and Lopardo (2012). Figure 9. Skewness pressure distribution along hydraulic jump. Adapted from Neto & Marques (2008), and Lopardo (2012). before the position $(x - x_1)/(y_2 - y_1) \approx 5$, i.e., the values tend to be higher than the mean value. This may be also extracted from Figure 4, in which instantaneous values at the roller region tend to be higher than the curve representing the mean depth + 3 times the standard deviation. The maximum skewness values were observed near the jump toe. In the downstream direction, the skewness values stabilize around zero. This trend is similar to data of Neto & Marques (2008), and Lopardo (2012) for pressure measurements, as shown in Figure 9. However, downstream the position $(x - x_1)/(y_2 - y_1) \approx 4$, the skewness values attain a minimum negative value around the position $(x - x_1)/(y_2 - y_1) \approx 6$, tending then to zero downstream of $(x - x_1)/(y_2 - y_1) \approx 8$. As discussed by Lopardo & Casado (2007), the transition from positive to negative skewness values seems to imply the possibility of a boundary layer separation from the stilling basin bottom. The present results show that there are remarkable similarities between the general behavior of the statistical variables associated to depth and pressure measurements in hydraulic jumps. When considering details as the length of concavity regions and the presence of critical points (minima, for example), differences were also observed. The similarities are related to the same cause of both fluctuations (pressure and depth), that is, the turbulent movement of the flow in the hydraulic jump. The differences follow from the different nature of the physical parameters

being analyzed, which point that more studies are nec

essary to verify possible quantitative relations between

them. A goal for such a relationship would be to use

one of the measurements as indirect “quantification”

of the second. Results of laboratory flumes were used

in the present study, involving small dimensions and

flow rates. The observed similarities suggest to extend the study to larger dimensions, so that eventual scale effects may also be also evidenced.

4 CONCLUSIONS

Ultrasonic sensors have been employed in hydraulic jumps for measurement of flow depths. It has proved to be a simple and promising technique for studies of strongly turbulent open flows. Because the flow depth fluctuations and pressure fluctuations are caused by the same turbulent condition, it is expected they are also related to each other. In the present study, results of experiments conducted for different inflow Froude numbers and upstream conditions (broad crested weir and sluice gate), using ultrasonic sensors, were pre sented. The sensors were moved along the central line of the experimental flumes, and were used to provide statistical information about the surface evolution. Sta tistical parameters of the flow depths (mean values, standard deviation, skewness) were compared to sta tistical values of pressure fields obtained from the literature.

The statistical treatment of the pressure fluctuations in the literature showed results having behaviors simi lar to that of the flow depths.The values of the standard deviations of both fluctuations (depth and pressure) reached their maxima in the region of the jump roller, decaying then to a reasonable constant value after wards, when the flow turbulence is less intense, and the flow stablishes the subcritical condition.

The present results involve relatively small dimen sions and flow rates, but the mentioned similarities point to the convenience of more studies for larger dimensions, evidencing scale effects, considering that phenomena of different natures are involved in the data for depths and for pressure.

This study also allows to suggest that the length of hydraulic jumps (and eventually also of the rollers) may be quantified based on the decay of the fluc tuations of the flow depths or the pressures. These characteristic lengths are of difficult measurement by visual observation, pointing to the necessity of quanti tative criteria for them, which depend on the equipment used to obtain the experimental data.

SYMBOLS

C'_y, C'_p Fluctuation coefficient of water depth and pressure data at location x, respectively

L_r Roller length

L_j Hydraulic jump length

Hager, W.H. 1993. Classical hydraulic jump: free surface profile. Canadian Journal of Civil Engineering 20(3): 536-539.

Kucukali, S. & Chanson, H. 2008. Turbulence measure ments in the bubbly flow region of hydraulic jumps.

Experimental Thermal and Fluid Science 33: 41–53.

Lennon, J.M. & Hill, D.F. 2006. Particle image velocimetry

measurements of undular and hydraulic jumps. Journal of

Hydraulic Engineering 132(12): 1283–1294.

Lin, C., Hsieh, S-C., Lin, I-J., Chang, K-A. & Raikar, R. 2012.

Laboratory measurements of a steady breaker using PIV

and BIV. Coastal Engineering Proceedings 33: 1–9.

Liu, M., Rajaratnam, N. & Zhu, D.Z. 2004. Turbulence struc

ture of hydraulic jumps of low Froude numbers. Journal

of Hydraulic Engineering 130(6): 511–520.

Lopardo, R.A. 2012. Internal flow of free hydraulic jump

in stilling basins. In 4th IAHR International Sympo

sium on Hydraulic Structures, 9–11 February 2012. Porto

Portugal: IAHR.

Lopardo, R.A. & Casado, J.M. 2007. Boundary layer sep

aration beneath submerged jump flows. In XXXII IAHR

Congress. Venice, Italy.

Marques, M.G., Drapeau, J. & Verrette, J-L. 1997. Flutuação

de pressão em um ressalto hidráulico. Revista Brasileira

de Recursos Hídricos 2(2): 45–52. [in Portuguese]

Marques, M.G., Gomes, J.F. & Endres, L.A.M. 2000.

Oscilação da posição inicial do ressalto hidráulico e o

campo de pressões na soleira de bacias de dissipação.

In Simpósio de Recursos Hídricos do Nordeste, 21-24 November 2000. Natal, Brazil: ABRH [in Portuguese]

Mignot, E. & Cienfuegos, R. 2011. Spatial evolution of turbulence characteristics in weak hydraulic jumps. Journal of Hydraulic Research 49(2): 222–230.

Murzyn, F. & Chanson, H. 2008. Experimental assessment of scale effects affecting two-phase flow properties in hydraulic jumps. Experiments in Fluids 45(3): 513–521.

Murzyn, F. & Chanson, H. 2009. Free-surface fluctuations in hydraulic jumps: Experimental observations. Experimental Thermal and Fluid Science 33: 1055–1064.

Murzyn, F., Mouzé, D. & Chaplin, J.R. 2007. Air-water interface dynamic and free surface features in hydraulic jumps. Journal of Hydraulic Research 45(5): 679–685.

Neto, E.F.T.N. & Marques, M.G. 2008. Análise do campo de pressões em ressalto hidráulico submergido a jusante de uma comporta. Revista Brasileira de Recursos Hídricos 13(4): 55–68. [in Portuguese]

Nóbrega, J.D. 2014. Metodologia teórica e experimental para determinação das características do ressalto hidráulico clássico, MScDissertation, School of Engineering at São Carlos, University of São Paulo, Brazil, Available from: http://www.teses.usp.br/teses/disponiveis/18/18138/tde 30092014-105847/pt-br.php. [in Portuguese]

Nóbrega, J.D., Schulz, H.E. & Zhu, D.Z. 2014. Free surface detection in hydraulic jumps through image analysis and ultrasonic sensor measurements. In Hubert Chanson and Luke Toombes (ed.), Hydraulic structures and society – Engineering challenges and extremes. 5th IAHR International Symposium on Hydraulic Structures, 25–27 June 2014. Brisbane Australia, doi:10.14264/uql.2014.42. Schulz, H.E., Nóbrega, J.D., Simões, A.L.A., Schulz, H. & Porto, R.M. 2015. Details of Hydraulic Jumps for Design Criteria of Hydraulic Structures, HydrodynamicsConcepts and Experiments, Schulz, H.E. (ed.), InTech, Available from: http://www.intechopen.com/books/hydro

Experimental study of head loss through an angled fish protection system

H. Boettcher, R. Gabl, S. Ritsch & M. Aufleger

Unit of Hydraulic Engineering, University of Innsbruck, Austria

ABSTRACT: This paper contains the results of an experimental study on head loss through angled screens for fish protection. A physical model on a scale of 1:2 was conducted to assess the effect of the geometry, defined by the angle, bar shape and bar spacing on head loss for a range of flow velocities. The results indicate that head loss increases with increasing blockage ratio, angle to the screen plane and the approach flow velocity, as expected by literature. Furthermore the measured data was compared with estimated results of some selected literature approaches, where their conformance is discussed in the paper.

1 INTRODUCTION

Hydropower plants delay or hinder downstream fish migration. In order to prevent fish from entering the turbine intakes, physical barriers, like fine screens with horizontal or vertical bars, bar spacings ≤20 mm are recommended. In general, these screens should be inclined or angled in the flow direction to guide fish to a bypass at the downstream end of the barrier (Larinier & Travade 2002; Dumont, 2013; Forum Fischschutz und Fischabstieg 2013). A conflict can arise due to difficult operational conditions as well as a strong increase in energy losses due to the high blockage ratio of the barrier.

A current research project at the University of Inns bruck is intended to advance a recently developed fish protection concept, with emphasis on operational, technical and biological issues. The ‘flexible fish fence’ is a physical barrier, created by horizontally arranged steel cables (Brinkmeier et al. 2013). The structure is positioned upstream of the turbine intakes, in a slight angle to the flow direction to guide fish to a bypass at the downstream end of the barrier.

This paper contains the preliminary results of an experimental study on head loss through the flexible fish fence. The paper focusses on the impact of geo metric (e.g. shape of steel cables, bar spacing, angle of screen plane) and hydraulic boundary conditions (e.g. approach velocity) on head loss.A physical model with a scale of 1:2 was designed at the hydraulic labora tory of the University Innsbruck. For the preliminary study, a structure similar to the flexible fish fence was installed in a rectangular flume. In order to exclude the effect of vibrations and roughness of the cable struc ture on head loss, rigid cylindric steel bars was used instead of steel cables. Overall nine screen configu rations, varying in their orientation angle (α) and bar spacing (b), were studied for a range of flow velocities. 2 LITERATURE REVIEW Various studies in the past have covered the issue of head loss at hydropower intake structures, thus several formulas have been proposed to calculate head loss at different kind of screens and thrash racks. Early studies were done by Kirschmer (1926), who found that bar size, shape and bar spacing of the thrash rack all influence the local head loss. According to his results, head loss Δh is given by where k_F is the bar shape coefficient, s the bar width, b the bar spacing and α the angle to the horizontal. Meusburger (2002) studied the influence of the blockage ratio due to the screen structure, the blockage ratio resulting from debris or trash, for example, and of different flow conditions on head loss and adjusted existing calculation approaches (Kirschmer 1926; Spangler 1929; Zimmermann 1969; Idel'cik 1979). Based on his results, the head loss coefficient ξ_M is given by with ξ_P , the head loss coefficient resulting from blockage, which varies with the blockage ratio p and the bar shape coefficient k_F as follows: where A_b is defined by the area of bars, A_s by the area through supports, spacers, etc. and A_t by the total area of the thrash rack in the flow. K_δ is a factor consid ering the approach flow angle δ. By multiplying with k_V the blockage by clogging and k_α the screen incli nation with $k_\alpha = \sin(\alpha)$ is considered. However, most studies dealing with the impact of screen inclination on head loss focussed on vertical bar configurations.

It is assumed, that k_α can be transferred to screens with horizontal orientated bars, angled to the flow direction. Raynal et al. (2013a, 2013b and 2014) investi gated different 'fish-friendly' trash rack configura tions including vertical angled trash racks with angled slats (bar racks), vertical angled trash racks with slats parallel to the flow direction (streamwise bar racks) and inclined trash racks in an experimental study. Thereby, Raynal et al. (2013a) addressed inclined screen configurations for a wide range of angles ($\alpha = 15^\circ$–90°) and bar spacings b ($b \leq 30$ mm), which is supposed to be most similar to the screens stud ied in the experiments presented in this paper. They considered the blockage ratio due to the bars and due to transverse elements (e.g. spacers) separately and proposed a modified factor k_α with $k_\alpha = \sin^2(\alpha)$, where

with p_b, the blockage ratio due to bars and supporting structures, p_{sp} the blockage ratio due to transversal elements (e.g. spacers) and C the shape coefficient of the transversal elements.

Other test setups for example used by Clark et al. (2010) or Kriewitz (2015) concentrated on submerged trash racks, louver and bar racks with different bar geometries, bar spacings and approach flow angles.

This paper presents the results for angled screens with horizontal arranged cylindric steel bars, similar to the concept of the flexible fish fence. The effects of the blockage ratio p and the angle α on head loss are studied within a range of different approach velocities. The results are then compared with the existing equa tions from Kirschmer (1926), Meusburger (2002) and Raynal et al. (2013a).

3 EXPERIMENTAL SETUP

The experiments were conducted in a rectangular flume, which is 20 m long, 0.8 m wide with a smooth bottom and side walls (Fig. 1). The channel slope was zero for all experimental runs. The downstream water depth was held constant to 0.4 m for the whole discharge range by a tailgate at the channel outlet. This paper focusses on the preliminary part of the study, which investigated the effect of the angle of ori entation α and bar spacing b. Therefore, nine screen configurations with horizontally placed cylindric bar rods with a diameter of s = 5 mm, afixed to two sup porting structures at the sides (Fig. 2, Table 1) were investigated. Additional vertical supports were inten tionally avoided in the pre-planning process, because

Figure 1. Scheme of the experimental channel – Flow direction from left to right; I = inlet, F = flow straightener, 1–8 = ultrasonic sensors, S = screen, T = tailgate, O = outflow Figure 2. Angled screen configuration: $\alpha = 45^\circ$, b = 5 mm (p = 0.5) – Flow

direction from the bottom to the top Table 1. Geometry of the screen configurations. b s p b/s α L n mm mm [-] [-] [∘] [m] [-] Model 5-15 5 0.25-0.5 1-3 30-90 0.8-1.5 25-50 Nature 10-30 10 0.25-0.5 1-3 30-90 1.6-3 50-100 the geometry should be similar to the flexible fish fence. However preliminary tests showed strong vibrations of the bars.Thus, additional vertical spacers were designed to reduce the vibrations (Fig. 2). All components were on a scale of 1:2, considering the ratio $h/s \geq 60$, which ensures that the resistance of the bars does not depend on the Froude number (Zimmermann 1969). Overall nine configurations with three different bar spacings (b) = 5, 10 and 15 mm with blockage ratios p of 0.25, 0.33 and 0.50 according to the definition of Meusburger (2002) and three angles (α) = 30, 45 and 90 ∘ were investigated (Fig. 3).The screens were placed 9 m downstream from the channel inlet (Fig. 1). According to the various bar spacings and angles, number of the bars (n) and bar respectively rack length varied (Table 1). Discharges (Q), which varied from 50 to 200 l/s and were measured by an electromagnetic flow sensor. The corresponding approach velocities ranged from 0.2 to 0.6 m/s (Tab. 1). Eight ultrasonic sensors measured the water depth upstream (h_1) and downstream (h_2) of the screen with an accuracy of ±1 mm. The pressure head loss was measured with two piezometer taps and a differential pressure transmitter with a

Figure 3. Layout of the studied angles of screen plane.

higher accuracy (±0.2 mm) between the upstream and downstream section of the screen. The upstream piezometer tab was located 3 m upstream of the barrier, the distance between the screen and the downstream measuring point was 6.5 m. They are considered to be out of backwater and drop curves of the screen and the gate at the outlet of the chan nel. Given the nine screen configurations and six discharges, in total 54 experiments with different con ditions were accomplished. Additional experiments were conducted to determine the head loss Δh_0 due to surface friction and the supports without the bar rods. The head loss Δh and head loss coefficient ξ was determined according

with the upstream flow velocity v_1, downstream velocity v_2, upstream water level h_1, downstream water level h_2, head loss Δh, head loss through friction and supports Δh_0 and head loss coefficient ξ.

4 RESULTS AND DISCUSSION

4.1 Influence of the model scale

Within the experiments, Reynolds numbers varied from Re = 750 to 3000, whereby Re is related to the bar thickness with s_{model} = 5 mm. Froude numbers ranged from Fr = 0.08 to 0.3. According to literature (Blevins 1984; Naudascher 1992), the resistance coefficient of cylindric cylinders is independent of the Reynolds number as long as [500 < Re > 20000], which covers the model conditions as well as nature conditions at hydropower intakes. Hence, the model scale of 1:2 is expected not to influence the results and the transferability to nature conditions is justified. Figure 4 shows the head loss coefficient ξ versus Reynolds numbers for selected experiments, confirming that ξ stayed roughly constant within the studied Reynolds number range. The frictional head loss and local head loss through the supports were removed for ξ.

4.2 Head loss

Figure 5 shows the head loss Δh versus Reynolds num

ber for the highest and lowest blockage ratio ($p = 0.50$
Figure 4. Independence of head loss coefficient ξ [−] of Reynolds Number Re [−] – including the mean value as dashed lines Figure 5. Head loss Δh versus Reynolds Number Re [−] for 6 configurations ($p = 0.25$, 0.5 and $\alpha = 30, 45, 90^\circ$). and $p = 0.25$) as well as the measured Δh_0 without the cylindric bar rods for $\alpha = 30^\circ$. As expected, Δh increases with the Reynolds number Re and the approach flow velocity. Furthermore, Δh increases significantly with a higher blockage ratio p. Obviously, the head loss through the supports and friction Δh_0 is relatively high considering the results for Δh with the lowest blockage ratio $p = 0.25$. This is mainly due to the two supports at the channel side walls, which constrained the cross section, resulting in flow separation. As suggested by the literature (Naudascher 1992), Δh_0 was slightly lower for the perpendicular configuration than for the two angled configurations. The effects of the support structure on the results were removed since the flexible fish fence has no supports. Besides, the results show that Δh for the configuration $p = 0.50$ and $\alpha = 30^\circ$ and $p = 0.25$ and $\alpha = 90^\circ$ are in a similar range, where Δh is slightly lower for the lower p. Figure 6 and Figure 7 demonstrate the effect of p and α on head loss coefficient ξ separately. It increases disproportionally with an increasing blockage ratio p, which is consistent with past studies (Kirschmer 1926;

Figure 6. Head loss coefficient ζ for $\alpha = 30, 45, 90^\circ$ in

function of the blockage ratio p [−]

Figure 7. Head loss coefficient ζ for $p = 0.25, 0.33, 0.5$ in

function of the angle α [$^\circ$]

Meusburger 2002; Raynal et al. 2013a). Thereby, ξ

increases from $p = 0.25$ to $p = 0.33$ by a factor of

1.7, whereas from $p = 0.33$ to 0.50 by a factor up to

2.5. Furthermore, the results show that ξ is a function

of α (Fig. 7) as expected by previous investigations

(Kirschmer 1926; Meusburger 2002; Raynal 2013a).

Head loss coefficients increased from $\alpha = 30^\circ$ to 45°

by a factor of up to 1.7 and from $\alpha = 45^\circ$ to 90° by a

factor of 2.1. Figure 7 suggests an almost linear behav

ior, but more angles have to be investigated to give a

significant tendency.A decrease of α is directly related

to an increase of bar length (L) respectively flow area

of the trash rack. This might have a significant effect

on the velocity profiles along the trash rack and there

fore affects the head loss coefficient.An ongoing study

on the structure of flow around the barrier will provide

a more detailed understanding. Head loss coefficients

for all nine screen configurations are summarized in

Table 2.

4.3 Comparative analysis

The head loss coefficient ξ was calculated by using the

equations of Kirschmer (1926), Meusburger (2002) Table 2. Head loss coefficients ζ for all nine different screen configurations. α [$\circ$] ξ [−] 30 45 90 p [−] 0.25 0.14 0.21 0.47 0.33 0.25 0.36 0.76 0.50 0.53 0.89 1.90 and Raynal et al. (2013a) and compared with the measured results. Figure 8 shows ξ versus α of the data for $p = 0.33$ and $p = 0.5$ and the estimated values from the current study.Thereby values from literature are shown for a range of α, which exceeds the range from the current data set, in order to see their trend. It is noticeable, that the values derived from Meusburger (2002) and Kirschmer (1926) overestimated the results for lower angles $\alpha = 30^\circ$ and 45° and the highest blockage ratio $p = 0.5$, whereas Raynal et al. (2013a) produced similar results compared to the data (deviations $\leq 7\%$ to the measured data). For the case of $\alpha = 90^\circ$ and $p = 0.5$ all estimated ξ are similar as a natural consequence considering ξ the formulas. For $\alpha = 90^\circ$ and lower blockage ratios the estimated values underpredicted the measured data slightly in a range from 7 to 26%.Again, for $p = 0.33$ the equation of Raynal et al. (2013a) produced the best results, with a slight tendency of underestimation. Figure 9 demonstrates the results and estimated ξ coefficients for an angle of $\alpha = 30^\circ$ and 45° plotted on the blockage ratio p. It illustrates again the high overestimation of ξ by Kirschmer (1926) and Meusburger (2002) especially for $p = 0.5$, where the deviations ranges from 41 to 68% compared to the measured results. The ξ -values derived from Raynal's equation slightly underestimated for $p = 0.25$ and 0.33, but delivered the best results for $\alpha = 30^\circ$ and 45°. The major differences of Raynal et al. (2013a) and Meusburger's equation is the separation between the blockage ratio through bars and transverse elements, the exponent of the head loss coefficient resulting from blockage p (ξ_P) was 1.65 instead of 1.5 and k_α with $\sin^2(\alpha)$ instead of $\sin(\alpha)$.The part of Raynal's equation considering the transverse elements can be ignored due to the test setup. Hence, ξ_P and k_α from Raynal's approach might be the factor that provided a better approximation for the studied screen geometries then Kirschmer (1926) and Meusburger (2002). Considering lower blockage ratios $p < 0.5$, the influence of the exponent in the term ξ_P is comparable low, with slightly lower results for ξ_P according to Raynal et al. (2013a) than ξ_P from Meusburger's equation. Whereas the coefficient $k_\alpha = \sin^2(\alpha)$ proposed by Raynal et al. (2013a) has a stronger effect on ξ, particularly for angles $\alpha \leq 45^\circ$. Thereby the modified k_α seems to be the main reason for the better results compared with measured data. The generally slight underestimation of ξ -coefficients, particularly for $p = 0.5$, estimated by Raynal

Figure 8. Head loss coefficient ξ measured and estimated by the equations of Kirschmer (1926), Meusburger (2002) and Raynal et al. (2013a) plotted on angle α [$^\circ$] for $p = 0.5$ and 0.33°.

Figure 9. Head loss coefficient measured and estimated by the equations of Kirschmer (1926), Meusburger (2002) and Raynal et al. (2013a) plotted on the blockage ratio p [–] for $\alpha = 30, 45^\circ$.

might be result from a different hydraulic behavior considering the cylindric bar shape in interaction with small bar spacings. During the experiments high fre quently oscillations and downstream turbulence par ticularly for high blockage ratios and higher Reynolds numbers were observed. Compared to a trash rack, studied by Raynal et al. (2013a) and other authors, the cylindrical bars with less or no bracing or spacers are very vunerable to vortex-induced vibrations.A numer ical study on rectangular bar profiles of Meusburger et al. (1999) showed that the progress of vortex sep aration is dependent on bar spacing, They found that the vortices influence each other from $b/s = 3$, which is the highest ratio for in the current test set up. With $b/s \leq 1.5$, vortices are strongly compressed, building a thick shear layer, which leads to an increase in flow velocity and head loss (Meusburger et al. 1999). Moreover, the frequency of vortex-induced vibra

Clark, S., Tsikata, J. & Haresighn, M. (2010). Experimental study of energy loss through submerged trahracks. Journal of Hydraulic Research. 48(1):113–118.

Dumont, U. (2013). Zum Stand der Technik einer ökologisch angepassten Wassernutzung. Seminar gewässerverträgli cher Wasserkraftausbau / Fischwanderhilfen, Renexpo.

Forum Fischschutz und Fischabstieg (2013): 4. Workshop –

Diskussionspapier. Im Auftrag des Umweltbundesamtes.

Idel'chik, I.E. (1979). Handbook of Hydrauic Resistance. (S. Edition, ed.)Hemisphere Publishing Corporation, New York, NY.

Kirschmer, O. (1926). Untersuchungen über den Verlust an Rechen, Mitteilungen Hydraulisches Institut München Nr.1.

Kriewitz, C.R. (2015). Leitrechen an Fischabstiegsanla gen: Hydraulik und fischbiologische Effizienz. VAW Mitteilungen (230).

Larinier, M. & Travade F. (2002). Downstream migration: problems and facilities. Bull. Fr. Peche Piscic. 364: 181–207.

Meusburger, H, Speerli, J. & Volkart, P. (1999). Untersuchun gen über den Einfluss der Geometrie und Anströmung von Einlaufrechen auf den Betrieb vonWasserkraftwerken (unpublished). Schlussbericht der Versuchsanstalt für Wasserbau, Hydrologie und Glaziologie der ETH-Zürich (VAW) an die Vereinigung Deutscher Elektrizitätswerke (VDEW) (4109). Meusburger, H. (2002). Energieverluste an Einlaufrechen von Flusskraftwerken, VAW-Mitteilungen (179). Naudascher, E. (1992). Hydraulik der Gerinnebauwerke. Springer Verlag, Wien (Austria), New York (NY). Raynal, S., Chatellier, L., Courret, D., Larinier, M. & David, L. (2013a). An experimental study on fish-friendly trashracks – Part 1. Inclined trashracks, Journal of Hydraulic Research 51(1): 56–66. Raynal, S., Chatellier, L., Courret, D., Larinier, M., & David, L. (2013b). An experimental study on fishfriendly trashracks – Part 2. Angled trashracks, Journal of Hydraulic Research 51(1): 67–75.

Raynal, S., Chatellier, L., Courret, D., Larinier, M., & David, L. (2014). Streamwise bars in fish-friendly angled trashracks. Journal of Hydraulic Research 52(3): 426–431. Spangler, J. (1928). Untersuchung über denVerlust an Rechen bei schräger Zuströmung. Mitteilung des hydraulischen Instituts der TH München (H2) Tanida, Y., Okajima, A. & Watanabe, Y. L. (1973). Stability of Circular Cylinder Oscillating in Uniform Flow or in a Wake. Journal of Fluid Mechanics. 61(4) Zimmermann, J. (1969). Widerstand schräg angeströmter Rechengitter. Universität Fridericana Karlsruhe,TherodorRhebock-Flussbaulaboratiorium (157)
Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Scale effects for air-entraining vortices at pipe intake structures

K. Tas¸tan

Department of Civil Engineering, Gazi University, Ankara, Turkey

ABSTRACT

Intake structures are used to withdraw water from reservoirs and rivers for several purposes such as irri gation, energy production, industrial cooling etc. If the vertical distance between intake and the water surface (submergence of the intake) is smaller than a certain limit value, which is called critical submergence, air starts to enter the intake due to free-surface air-core vortices. Since the entrained air reduces the efficiency of the intake and may generate operation problems downstream, determination of critical submergence and the influencing parameters is critically important. Hydraulic engineers have generally used Froude simi larity between model and prototype intakes. However,

previous studies showed that the Froude similarity may not represent the air entrainment or critical submergence at prototype intakes. Yıldırım and Kocabaş (1995) introduced the concept of imaginary critical spherical sink surface (CSSS) for the prediction of critical submergence of pipe intakes located in open channel flow, by superposing potential point sink flow and uniform flow. In this study, it is shown that if the ratios of average pipe intake velocity to the velocity at the CSSS between the model and prototype pipe intakes are identical (kinematic similarity), dimensionless critical submergences (ratio of critical submergence to the intake diameter) of corresponding intakes become also identical. Kinematic similarity concept can be applied both for the intakes located in still-water reservoir or cross-flow. In cross-flow cases, since the velocity at the CSSS is equal to the $U_\infty/2$ (if the intake is not affected by the friction caused by a boundary close to the free-surface or imposed circulation), pipe intakes having identical V/U_∞ also have identical S_c/D (in which, U_∞ is the average velocity of cross-flow, V is the average velocity at the intake, S_c is the critical submergence, and D is the diameter of the intake). Table 1 shows that kinematic similarity should be used instead of Froude similarity (F is intake

Froude number).

In the case of still-water reservoir, the velocity at the CSSS cannot be predetermined. Therefore, tests should be conducted to obtain the velocity at the CSSS.

Analysis of various Piano Key Weir geometries concerning discharge coefficient development

M. Oertel & F. Bremer

Hydraulic Engineering Section, Civil Engineering Department, Lübeck University of Applied Sciences, Lübeck, Germany

ABSTRACT: Piano Key Weirs (PKW) are relatively new hydraulic structures and research investigations start a couple of years ago. PKWs have modified the labyrinth weir geometry and guarantee an effective discharge, e.g. as flood release structure on a dam or as replacement of regular shape weirs in river systems. Current research focuses on discharge coefficients for various discharge amounts. For small discharges regular weirs are less effective than PKWs, hence replacements might be reasonable. Thereby, a couple of questions are in the main focus of interest, like e.g. general hydraulic phenomena, sediment transport, fish climb capability, or geometry adaption. Usually, PKWs are analyzed in scaled experimental models, while discharge coefficients can be calculated via Poleni or Du Buat approach. Additionally, numerical 3D CFD simulations are available to confirm experimental model results. The present paper deals with various laboratory-scaled PKW geometries and analyzes their discharge coefficients. Investigations are carried out within a 10 m long and 0.8 m wide tilting

flume. Upstream water levels are measured via ultrasonic sensor as longitudinal water surface profiles. The paper answers the question how three selected geometries will influence the PKW's discharge coefficients and their efficiency.

1 INTRODUCTION AND LITERATURE REVIEW

Piano Key Weirs (PKW) are relatively new hydraulic structures, which can be used as flood release struc tures for reservoir systems or as weir replacement in regular flow channels. PKWs are part of the non linear weir group. General designs were developed by Blanc & Lempérière (2001) and Lempérière & Ouamane (2003). Pralong et al. (2011) introduced a standard naming convention for PKWs. A major benefit of a PKW is an increased discharge capacity relative to linear weirs, and it is ideal for top-of-dam spillway control structure applications as the footprint space requirement is relatively small in comparison to the developed weir length geometry (Oertel & Tullis 2014). A significant amount of research has been carried out during the last years using laboratory scale physical models; e.g. Machiels et al. (2011a), Anderson & Tullis (2012) or Oertel (2015).

PKWs have been generally classified into four main geometric types: (1)TypeA, (2)Type B, (3)Type C, and

(4) Type D. Differences can be found concerning the

main geometry. A Type A PKW features symmetrical

keys relative to a transverse centerline axis.Type B fea

tures cantilevered apexes on the upstream end (outlet

keys) and vertical apex walls on the downstream end

(inlet keys). Type C is the opposite of Type B with the

cantilevered apexes on the downstream end. Type D is a rectangular labyrinth weir (vertical apex walls); see e.g. Oertel & Tullis (2014). Inlets demonstrate downstream keys, outlets upstream keys. One complete PKW unit will be described by one inlet, two side walls and a half outlet. General PKW parameters are (see also Fig. 1). Usually, PKW flow characteristics and discharge coefficients were experimentally analyzed within scaled physical models; e.g. Machiels et al. (2011a), Kabiri-Samani & Javaheri (2012) or Anderson (2011). Dabling & Tullis (2012) additionally compared their PKW results for submerged head discharge relationships with labyrinth weirs. Ribeiro et al. (2012) identified primary and secondary parameters, having a significant effect on the PKWTypeA discharge capacity. Anderson & Tullis (2012) evaluated PKW Type A head discharge behaviors for in-channel and reservoirapproach flow conditions. Oertel & Tullis (2014) compared experimentally determined discharge coefficients for several PWK types with those from a numerical 3D CFD model and found a good agreement and applicability of the VOF code. Oertel (2015) analyzed additional variations of PKW Type A geometries with semi-circle and triangle expansions. A basic investigation on PKW analyzing procedures is presented in Oertel (2016). Generally, discharge coefficients can be calculated by measuring the upstream water surface level h T (above weir crest) within experimental or numerical

Figure 1. Main geometric PKW notations, flow direction

left to right.

P i upstream weir height,

P o downstream weir height,

W i inlet key width,

W_o outlet key width,

N_u No. of PKW units,

B_h side crest length,

B_i downstream overhang length,

B_o upstream overhang length,

B_b weir foot length,

T_s wall thickness,

W_u PKW unit width, $W_u = W_i + W_o + 2T_s$,

W total weir width, $W = N_u \cdot W_u$,

L_u crest centerline length of PKW unit, $L_u = W_u + 2B_h - 2T_s$,

L total crest centerline length, $L = N_u \cdot L_u$.

models and calculating the associated velocity head

v_T averaged over total flow depth ($h_T + P$, where P is

the weir height) (Oertel 2016). Discharge coefficients

C_d can be determined subsequently by Poleni/Du Buat

formula:

where Q = discharge, C_d = dimensionless discharge

coefficient, C' = coefficient including 2/3 and C_d and

$(2g)^{0.5}$, L = total centerline crest length, g = accelera

tion due to gravity, H_T = total upstream energy head

including velocity head = $h_T + v_T^2 (2g)^{-1}$.

2 EXPERIMENTAL MODEL

2.1 General remarks

The present paper focuses on water surface profile, discharge coefficient and efficiency investigations of

three various PKW geometries with free overfall con

ditions: (1) Type A PKW, (2) Type B PKW, and (3)

Type A PKW with fractal components, based on Bre

mer (2015). Consequently, PKW geometries will be

labeled PKW_A, PKW_B, and PKW_A F . The main

aim of the present investigation is to identify discharge

coefficients for various PKW geometries and to com

pare these among each other as well as with values

from the literature. Figure 2. Schematic plan view on laboratory water circulation system. Table 1. Investigated PKW geometries. Parameter PKW_A PKW_B PKW_A F P [mm] 196.9 196.9 196.9 L [mm] 4667.2 3634.4 6691.2 W [mm] 800.0 800.0 800.0 W i [mm] 105.0 105.0 105.0 W o [mm] 84.0 84.0 84.0 B b [mm] 230.7 230.7 230.7 B h [mm] 488.9 359.8 741.9 B i [mm] 129.1 – 129.1 B o [mm] 129.1 129.1 129.1 T s [mm] 5.0 5.0 5.0 N u [–] 4 4 4 W i W -1 o [–] 1.25 1.25 1.25 PT -1 s [–] 39.38 39.38 39.38 LW -1 [–] 5.86 4.57 8.41 PT -1 s [–] 41.4 41.4 41.4 Physical modeling data were produced at Lübeck University of Applied Sciences’ Water Research Laboratory. Three PKW structures were installed within a tilting flume; length L = 10.0 m, width W = 0.8 m, height H = 0.8 m. Flow was supplied to tilting flume via piping containing magnetic inductive flow meter (MID, fabricate: Krohe, model: Optiflux 2000, accurate to ±0.1 l/s). Figure 2 gives a schematic plot. Flow depths were determined using ultrasonic probes (USS, fabricate: general acoustics, model: USS635, accurate to ±1 mm). An automatic step motor positioning system (fabricate: isel, accurate better than 1 mm) guarantees a precise measurement procedure. 2.2 PKW geometries The main geometric parameters for all three investigated PKW geometries are shown in Table 1. All PKWs are fabricated with equal inletand outlet key widths (W i = 105 mm, W o = 85 mm), representing a key width ratio of W i W -1 o = 1.25. Also weir heights P, weir widths W , weir foot lengths B b and number of PKW units N u are similar for all three investigated PKW geometries.

Figure 3. 3D view of investigated PKW geometries, flow direction left to right.

2.2.1 PKW_A

PKW_A geometry represents a symmetric structure with same downstream and upstream overhang lengths ($B_i = B_o = 129.1$ mm). The total crest center line length is $L = 4667.2$ mm with resulting length width ratio $LW^{-1} = 5.86$. Figure 3a shows a 3D view for the investigated PKW_A geometry.

2.2.2 PKW_B

Ouamane & Lempérière (2013) mention that a PKW model without downstream overhang is characterized by an increased efficiency compared to PKWs with downstream overhang (capacity increase of approx. 12% for relative head $hH^{-1} < 0.4$). Additionally the authors mentioned an increased efficiency of 7% for PKWs with symmetrical overhangs compared to those PKW having an asymmetrical overhang.

In the present investigation PKW_B geometry equals PKW_A upstream PKW's longitudinal center line. A major variation can be found for the downstream area since PKW_B exists of no downstream overhang length. Figure 3b shows a 3D view for the investigated PKW_B geometry.

2.2.3 PKW_A F

PKW_A F geometry is a further development of PKW_A. Therefore, fractal elements were included on the crest of PKW_A walls – see also Laugier et al.

(2012) and . Since the flow above the structure will

be guided banded into the outlet keys, fractal elements

might be placed with a defined angle in further investi

gations to increase resulting discharge capacities. The

present investigation focuses on five orthogonal frac

tal elements at each wall. Thereby, only an upstream

overhang length was chosen for fractal elements, rep

resenting a PKW_B structure within a PKW_A. Fig

ure 3c shows a 3D view for the investigated PKW_A F

geometry.

2.3 Model runs

For each PKW geometry 18 model runs, hence

54 runs in total, were performed with varying dis

charges (specific discharges q = 2.50, 6.25, 10.00,

13.75, 17.50, 21.25, 25.00, 31.25, 37.50, 43.75,

50.00, 56.25, 62.50, 75.00, 87.50, 100.00, 112.50,

125.00 $l(sm)^{-1}$). Measurements were conducted on Figure 4. Qualitatively flow pattern comparison for PKW_A (left) and PKW_A F (middle and right) geometries, left and middle: small discharges, right: large discharges. a total length of ~2.5 m upstream PKW longitudinal centerline on a 1 cm measurement grid. Hence, data of approximately 250 measurement points were collected. Measurement time was set to 10 seconds. Resulting data were filtered and corrected using a standard deviation criterion to remove outliers (outlier if: $d > m + 2s$ or $d < m - 2s$, where d = time dependent flow depth data point, m = mean time averaged flow depth, s = standard deviation). Finally, corrected time-averaged flow depths were used for data analysis. Upstream energy heads were calculated using the upstream flow depth h minus weir height P. Additionally, mean velocity head $v^2\ T\ (2g)^{-1}$ was added to fulfill Du Buat formula (Eq. 1). Hence, the total energy head becomes H T = h T + v 2 T (2g) −1 . Oertel (2016) mentions the sensitivity of measurements concerning chosen data points for result analysis and identifies a distance of 5 × P upstream PKW's crest centerline for save result production. Consequently, for the present investigation data analysis will be performed using USS data points at a distance of 5 × P for all model runs. More details for recommended data analysis procedures can be found in Oertel (2016). 3 RESULTS AND DISCUSSION 3.1 General remarks All model configurations will be analyzed concerning: (1) water surface profiles, (2) discharge coefficients, (3) PKW efficiency and (4) scale effects. Consequently, investigation results compare all PKW geometries and explain differences as well as possible advantages or disadvantages of such structures.

Figure 5. Exemplary WSL at flume's centerline for inves

tigated discharges, PKW_A (inlet key section, exaggerated

plot).

Figure 6. Direct comparison of water surface profiles for

PKW_A, PKW_B and PKW_A F geometries at measuring

distance 5 × P.

3.2 Water surface profiles

Figure 5 exemplary shows measured longitudinal

water surface profiles of all investigated discharges at

flume's centerline (inlet key) for PKW_A geometry.As

described previously water surface profiles are a result

of time averaged flow depth measurement data on lon

gitudinal axis. It can be shown that smooth water sur

faces will occur up to a distance of xP −1 > 2. Between

0 < xP −1 < 2 flow will be majorly accelerated and sur

face profiles becomes declined. For −1 < xP −1 < 0

resistance forces of structure's overflowing edges lead to increased and waved water surfaces. Fully three dimensional effects can be observed (see Figs. 4 and 10). Overflowing discharge will be redirected lateral to main flow direction and with increasing discharges an influence of opposite PKW units can be observed. Especially for PKW_A F scale effects may influence water surface profiles and resulting discharges coeffi cients for smaller discharges ($q < 15$ l(sm) $^{-1}$). Hence, further investigations are necessary to identify quanti tative influences. These can be arranged as large scale experimental or numerical model simulations. Within the present investigation scale effects will be neglected

concerning data analysis. Figure 7. Discharge coefficients C_d for PKW_A, PKW_B and PKW_A F geometries. Additionally, Fig. 6 gives comparable results for water surface levels at measuring point $5 \times P$. 3.3 Discharge coefficients Discharge coefficients C_d , calculated by use of Eq. 1, are shown as absolute values in Fig. 7a. The influence of structural design in terms of resulting discharge coefficients will be studied by using PKW_A with symmetric overhangs as reference geometry. Figure 7b shows the relative comparison of resulting C_d values for all investigated PKW geometries. It can be shown that largest C_d values can be determined for PKW_B for $H_T P^{-1} > 0.05$. Contrary, PKW_A F offers smallest C_d values for $H_T P^{-1} > 0.05$. Only for $H_T P^{-1} < 0.05$ the fractale geometry shows majorly increased values (maybe as a result of scaling effects). With increasing upstream energy heads ($H_T P^{-1} > 0.05$ to 0.07), hence, with increasing discharges, discharge coefficients will be decreased

Figure 8. Normalized discharge coefficients C_{dw} for PKW_A, PKW_B and PKW_A F geometries.

continuously. This effect was observed by all pre vious authors. Due to increased upstream velocity heads and water amounts, the discharge passes the weir more and more in longitudinal direction with less three-dimensional effects concerning the flume width (see Fig 10). Hence, using the total centerline crest length for C_d value calculation lead to smaller discharge coefficients since only the channel width becomes flow effective. As a result C_d values reach a minimum of 0.22 (PKW_A_F), 0.33 (PKW_A) and 0.37 (PKA_B) for investigated maximum discharges $q = 125.0\ l(sm)^{-1}$, as shown in Fig. 7 and Tables 2 to 4. It should be noted that discharge coefficient for Type A PKW geometries can also reach mini mum values less than 0.2 for very large discharges ($q \approx 256\ l(sm)^{-1}$, $H_T P^{-1} = 1.1$); see Anderson & Tullis (2012).

3.4 PKW efficiency

Since C_d values will be calculated by using the total centerline crest length, these values are not reasonable for adequate PKW efficiency statements. Therefor, a

direct comparison of resulting upstream flow depths Figure 9. Weber numbers We for PKW_A, PKW_B and PKW_A_F geometries. is necessary, showing an indication of structure’s efficiency (see Fig. 6). Another way to identify PKW efficiency is to use normalized discharge coefficients C_{dw}, as described subsequently. Fig. 6 shows resulting water surface profiles for all three investigated PKW geometries at measuring distance $5 \times P$. It can be shown that PKW_B lead to larger upstream flow depths for same specific discharges than PKW_A and PKW_A_F. Hence, PKW_B comes with less efficiency concerning discharge capacity. Precisely, analyzing the absolute PKW efficiency, normalized discharge coefficients C dw are necessary to allow result comparison between various PKW geometries with different total centerline crest lengths. Therefore, normalized C dw values are determined by implementing the fixed flume width: where C dw = normalized discharge coefficient, W = fixed flume width = 0.8 m (for present investigation). If the purpose of the weir is to produce a low upstream head the calculated C d values including the crest length L don’t allow a evaluation of the efficiency of the investigated weir geometry. The normalized C dw value becomes necessary to guarantee a direct efficiency comparison. See same procedure e.g. in Machiels et al. (2011b). Figure 8 shows resulting normalized discharge coefficients C dw for all investigated PKW geometries as absolute values and relative comparison. It can be found, in agreement with Fig. 5, that PKW_A and PKW_A F demonstrate best performances concerning their efficiency with comparable similar values for 0.12 < H T P -1 < 0.4. For H T P -1 < 0.1 PKW_A F shows maximum C dw values up to 7, which might be a result of scale effects combined with positive consequences of included fractal elements. For this region further investigations to confirm negligence of scale effects is necessary.

Figure 10. Exemplary photograph of experimental model runs, left: q = 17.5 l(sm) -1 , middle: q = 50.0 l(sm) -1 , right:

q = 125.0 l(sm) -1 . Table 2. Tabularized results for PKW_A geometry. Q q h T (at 5×P) v T H T C D C DW ls -1 l(sm) -1 cm ms -1 cm - - 2.00 2.50 4.51 0.012 4.51 0.478 2.789 5.00 6.25 6.92 0.030 6.92 0.624 3.643 8.00 10.00 9.14 0.047 9.14 0.652 3.802 11.00 13.75 11.10 0.064 11.10 0.664 3.872 14.00 17.50 12.76 0.081 12.76 0.678 3.955 17.00 21.25 14.40 0.098 14.40 0.679 3.963 20.00 25.00 16.30 0.114 16.30 0.657 3.833 25.00 31.25 19.68 0.140 19.68 0.610 3.557 30.00 37.50 23.39 0.166 23.39 0.558 3.254 35.00 43.75 26.40 0.191 26.40 0.535 3.120 40.00 50.00 29.78 0.215 29.78 0.504 2.940 45.00 56.25 32.88 0.238 32.89 0.482 2.814 50.00 62.50 36.54 0.261 36.54 0.453 2.645 60.00 75.00 42.82 0.305 42.82 0.420 2.449 70.00 87.50 49.05 0.347 49.05 0.392 2.286 80.00 100.00 56.35 0.386 56.36 0.359 2.095 90.00 112.50 61.17 0.426 61.17 0.350 2.039 100.00 125.00 67.26 0.463 67.27 0.332 1.937 Table 3. Tabularized results for PKW_B geometry. Q q h T (at 5×P) v T H T C D C DW ls -1 l(sm) -1 cm ms -1 cm - - 2.00 2.50 3.75 0.012 3.75 0.810 3.682 5.00 6.25 6.76 0.030 6.76 0.830 3.772 8.00 10.00 9.19 0.047 9.19 0.831 3.776 11.00 13.75 11.62 0.064 11.62 0.797 3.620 14.00 17.50 13.86

0.081 13.86 0.772 3.505 17.00 21.25 15.98 0.097 15.98 0.750 3.409 20.00 25.00 18.62 0.113 18.62 0.697 3.164 25.00 31.25 22.67 0.138 22.67 0.641 2.910 30.00 37.50 26.16 0.164 26.16 0.612 2.781 35.00 43.75 30.13 0.188 30.13 0.572 2.597 40.00 50.00 33.44 0.211 33.44 0.552 2.508 45.00 56.25 36.80 0.235 36.81 0.532 2.416 50.00 62.50 40.08 0.257 40.08 0.514 2.337 60.00 75.00 46.29 0.301 46.30 0.487 2.211 70.00 87.50 53.28 0.341 53.29 0.453 2.056 80.00 100.00 59.91 0.380 59.91 0.427 1.941 90.00 112.50 67.75 0.416 67.76 0.396 1.799 100.00 125.00 74.96 0.450 74.97 0.374 1.700 Table 4. Tabularized results for PKW_A F geometry. Q q h T (at 5×P) v T H T C D C DW ls -1 l(sm) -1 cm ms -1 cm – – 2.00 2.50 2.51 0.012 2.51 0.802 6.708 5.00 6.25 4.39 0.030 4.39 0.856 7.156 8.00 10.00 6.61 0.048 6.61 0.734 6.142 11.00 13.75 8.61 0.065 8.61 0.672 5.619 14.00 17.50 10.74 0.082 10.74 0.608 5.083 17.00 21.25 12.61 0.099 12.61 0.574 4.799 20.00 25.00 14.91 0.115 14.91 0.520 4.352 25.00 31.25 18.41 0.141 18.41 0.467 3.908 30.00 37.50 22.30 0.166 22.30 0.416 3.479 35.00 43.75 25.84 0.191 25.84 0.384 3.213 40.00 50.00 28.96 0.216 28.96 0.365 3.054 45.00 56.25 32.61 0.239 32.61 0.340 2.846 50.00 62.50 36.19 0.261 36.19 0.320 2.679 60.00 75.00 43.07 0.305 43.07 0.291 2.430 70.00 87.50 49.97 0.346 49.98 0.267 2.232 80.00 100.00 54.29 0.389 54.30 0.262 2.194 90.00 112.50 62.44 0.424 62.45 0.238 1.989 100.00 125.00 69.15 0.459 69.16 0.224 1.874

PKW_B almost reaches the other PKW’s efficiency

only for H T P -1 > 0.3, but without showing a benefit

for all investigated discharges. Comparing discharge

coefficients C d (Fig. 7) with normalized discharge

coefficients C dw (Fig. 8) shows a feasible wrong inter

pretation of PKW’s efficiency, because C d values

are larger for PKW_B than for all other investigated

geometries because the total cenerline crest length is

much smaller – but the efficiency definitely is less (see

C dw values and upstream water surface levels). Conse

quently, a sensitive data analysis is necessary to allow

significant research output for efficiency statements of PKW geometries. 3.5 Scale effects Since tested PKW geometries represent a scaling factor of approx. 1/10, scale effect (viscosity influence and surface tension) may occur and must be discussed concerning experimental model results and prototype transfer; see e.g. Heller (2011). Surface tension effects can be neglected by using a minimum flow depth (upstream head) of 20 mm (Bretschneider 1980) or 25 mm (Ettema 2000). For the special case of Piano Key Weirs Cicero et al. (2011) investigated a 1/30 scaled PKW for Malarce dam in France. The authors mentioned that especially for small discharges ($H_T P^{-1} < 1$) scale effects influence resulting

discharge coefficients up to 20% as a result of surface tension (Weber number). Another scale effect analysis comes by Erpicum et al. (2014b), where PKWs with geometric scale factors of 1/7, 1/15 and 1/25 were investigated; respectively, a design discharge of 10 $m^3 s^{-1}$ was reproduced by Froude similitude as 77.1, 11.5 and 3.2 ls^{-1}. Concluding, the authors mention a save use of scaled PKW models if the upstream head is higher than 60 mm (for analyzing flow conditions) or 30 mm (for discharge capacity analysis). Otherwise, surface tension will reduce air entrainment and decrease resulting discharge coefficients. Generally, the Weber number is a criterion to identify scale effects due to surface tension:

where ρ = water density, g = gravitation due to gravity, H = total energy head, σ = surface tension. Figure 9 gives Weber numbers for all investigated PKW geometries and discharges. It can be shown, that $We \leq 10$ for $H_T P^{-1} < 0.05$ and $We \leq 50$

for $H_T P^{-1} < 0.1$. Hence, for the present investigation results must be interpreted very carefully for small discharges $H_T P^{-1} < 0.07$, where $We \leq 30$, because surface tension may influence overtopping flow depths and discharge coefficients will be underestimated. In consequence, large scale experimental models are necessary to analyze this very sensitive flow situation for small $H_T P^{-1}$ ratios.

4 CONCLUSIONS

The present paper deals with various Piano Key Weir geometries and compares resulting water surface profiles and discharge coefficients concerning the structure's efficiency.

It could be shown, that upstream water surface profiles are covered by three-dimensional flow effects due to the PKW's special geometry with inlet and outlet keys. For the inlet key water surface profiles will be accelerated close to the weir's longitudinal centerline and decelerated due to occurring resistance forces at structure's overflowing edges, which results in a waved profile.

Discharge coefficients show a typical development with larger values for small discharges since the structure is more effective if a non-influenced fully three-dimensional flow occurs and the total centerline crest length becomes flow effective. For increasing discharges the flow changes with two dimensional characteristics and discharge coefficients will be reduced.

Analyzing and comparing discharge coefficients C_d and normalized discharge coefficients C_{dw} con firms non-practicability of C_d values for efficiency statements because these values are calculated with included centerline crest lengths. Since these lengths

characteristics over a Piano Key Weir. J. Hydr. Res., 49(3), 359–366.

Machiels, O., Erpicum, S., Archambeau, P., Dewals, B. & Pirotton, M. (2011b). Piano Key Weir preliminary design method – Application to a new dam project. Proc. 1st Workshop on Labyrinth and Piano Key Weirs, PKW 2011, Taylor & Francis Group London, 199–206.

Oertel, M. & Tullis, B.-P. (2014). Comparison of Piano Key Weir Discharge Coefficients from experimental and numerical models. Proc. 3rd IAHR Europe Congress, Porto, Portugal.

Oertel, M. (2015). Discharge Coefficients of Piano KeyWeirs from Experimental and Numerical Models. Proc. 36th IAHR World Congress, The Hague, the Netherlands.

Oertel, M. (2016). Sensitivity Analysis for discharge coef ficients of Piano Key Weirs. 6th Intern. Symposium on

Hydraulic Structures, IAHR, Portland, USA, accepted

abstract, accepted paper. Ouamane, A. & Lempérière, F. (2013). PK Weir, Design of a New Economic Shape Of Weir Dams, Piano Keys Weirs, Tidal Energy & Energy Storage., http://
www.hydrocoop.org/pk-weir-design-neweconomic-shape-of-weir. Pralong, J., Vermeulen, J., Blancher, B., Laugier, F., Erpicum, S., Machiels, O., Pirotton, M., Boillat, J. L., Leite Ribeiro, M. & Schleiss, A. J. (2011). A naming convention for the Piano Key weirs geometrical parameters. Proc. Intl. Conf. Labyrinth and Piano Key Weirs, Liege, Belgium. Ribeiro, M. L., Pfister, M., Schleiss, A. & Boillat, J.-L. (2012). Hydraulic design of A-type Piano Key Weirs. J. Hydr. Res., 50(4), 400–408. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Determination of discharge coefficient of triangular labyrinth side weirs with one and two cycles using the nonlinear PLS method

M.A. Nekooie

Department of Emergency Management, Malek Ashtar University of Technology, Tehran, Iran

A. Parvaneh

Department of Civil Engineering, Sharif University of Technology, Tehran, Iran

A. Kabiri-Samani

Department of Civil Engineering, Isfahan University of Technology, Isfahan, Iran

ABSTRACT: Side weirs are hydraulic control structures widely used in irrigation, drainage networks and

waste water treatment plants. These structures can be used for adjusting and diverting of flow with minimum

energy loss. In spite of many studies were carried out on rectangular side weirs, the studies on oblique and

labyrinth side weirs are scarce. In this study, based on the experimental data from more than 210 laboratory tests and through using the multivariable nonlinear partial least square (PLS) method, two nonlinear equations are presented for discharge coefficient C_M of triangular labyrinth side weirs with one and two cycles. The obtained empirical equations relating C_M with the relevant geometric and hydraulic dimensionless parameters L'/B, L'/L, $Fr_1/\sin(\delta/2)$ and $p \cdot \sin(\delta/2)/(h_1 - p)$ in a rectangular open channel. Comparison between results of the new presented equations and the measured data shows that the proposed empirical equations can predict the discharge of diverted flow over side weirs with good accuracy.

1 INTRODUCTION

Irrigation techniques are absolutely depended on the application of the side weirs. Besides it, these structures are normally used for water networks and even flood protection to control the overwhelmed water (Hager 1987; and Fritz and Hager 1998).

According to the themes of the succeeding investigations on side weirs, the studies can be classified into four major categories.A number of former studies have focused on identifying the side weirs' discharge coefficient with suband/or supercritical approach flow (Azimi et al. 2014 and Crispino et al. 2015). The second group have focused on a progressing subject, e.g. increasing the efficiency of the side weirs, by applying the modified shapes including oblique, triangular, trapezoidal, elliptical and labyrinth side weirs (Parvaneh and Borghei 2009; Parvaneh et al. 2010; Borghei and Parvaneh 2011; Borghei et al. 2013; Emiroglu et al. 2014; Nezami et al. 2015; Aydin and Emiroglu 2016; Aydin and Kayisli 2016; and Parvaneh et al. 2016). Some others have commonly focused on the effects of the main channel cross-section (e.g. trapezoidal, circular, parabolic or triangular channels) on the efficiency of the side weirs of different geometries (Cheong 1991; Uyumaz 1992; Vatankhah 2012; Vatankhah 2013; and Azimi and Shabanlou 2015). More recent studies have simulated/predicted the side weir discharge coefficient applying computational fluid dynamics and soft computing methods (Emiroglu et al. 2010;Aydin 2012; Vatankhah 2012; Aydin 2015; Parsaie and Haghiabi 2015; and Zaji et al. 2016). Most of the aforementioned studies were subjected to determine the discharge coefficient of side weirs. In practice, the side weir discharge coefficient is a major factor to be considered. For a given opening length along the side-wall of the main channel, applying the modified shapes such as; triangular and asymmetric labyrinth side weirs results in the increase of the discharge coefficient. Former studies indicated that the angle of flow diversion due to the use of oblique side weirs γ and vertex angle of triangular and trapezoidal side weirs δ along with the upstream approach flow Froude number Fr_1 significantly affect the side weir discharge coefficient (Ura et al. 2001; Parvaneh 2008; Borghei and Parvaneh 2011; Parvaneh et al. 2012; Borghei et al. 2013; Emiroglu et al. 2014; Parvaneh et al. 2016). Until quite recently, approximate methods based on experiments conducted over a limited range of the many variables involved have been used. In many cases, the use of such methods caused very substantial errors in the calculated spill discharge. Partial least square method PLS, artificial neural networks ANN, fuzzy logic FL and adaptive neuro-fuzzy inference system ANFIS methods are recently widely used for determination of side weir flow discharge coefficient (Ramamurthy et al. 2006; Aydin 2015; and Bonakdari et al. 2015). Borghei and Parvaneh (2011),

Figure 1. Schematic view of the triangular labyrinth side weir; a) plan, b) section.

and Borghei et al. (2013) showed that efficiency of the labyrinth side weirs could be improved by apply ing a modified shape of oblique-triangular labyrinth side weir. The main objective of the present study is to develop a more precise semi-analytical approach for determination of the discharge coefficient of modified oblique-triangular labyrinth side weirs using nonlinear PLS method.

Borghei et al. (2013) studied triangular labyrinth side weirs (Fig. 1). They conducted over 200 experi mental tests (see Table 1) and introduced Eqs. (1) and (2) for estimation of the De Marchi coefficient of dis charge C_M for triangular labyrinth side weirs with one and two cycles respectively.

Where C_M = De Marchi coefficient of discharge (-); B = channel width (m); L = opening length of side weir (m); L' = weir crest length (m); p = weir height (m); δ = vertex angle (degrees); h_1 = flow depth at the upstream end of the side weir (m); and Fr_1 = Froude number at the upstream end (-).

The current study also aims to contribute to the literature by introducing new equations for dis charge coefficient C_M of triangular labyrinth side

weirs through using the multivariable nonlinear par

tial least square (PLS) method. Finally the accuracy of presented equations will be evaluated using the experimental data and corresponding equations in the literature. 2 DATA ANALYSIS Considering Fig. 1, the De-Marchi approach, by assuming constant energy in the main channel, along the side weir length, leads to the following equations: In which ▮(h, E, p) is; Where "s", "B", "p" and "h" are the flow direction, channel width, side weir height and water depth respectively as showed in Fig. 1. "E" is the specific energy and "C_M" is the discharge coefficient of the side weir. Thus, the relation between C_M and other hydraulic variables of the flow would be; Where, 1 and 2 are accounted for immediately upstream and downstream of the weir respectively, and C_M has to be found experimentally. In the conducted tests, one of the important variables is weir effective length determined as follows, Eqs. (6). Beside the popular dimensionless parameters of normal side weirs, i.e., Fr_1 and p/h_1, other variables such as δ and L'/L were considered for the triangular labyrinth side weirs. No doubt that with many different functions of these variables, a wide range of combination in the form of equation could be obtained. Therefore based on the results of the data analysis, a nonlinear relation is presented for discharge coefficient in terms of geometric and hydraulic parameters through using the partial least square (PLS) method. 3 APPLICATION OF PLS METHOD Partial least square (PLS) is a robust method to estimate and fit multivariable statistical data. In effect, PLS is used to determine a dependent variable in terms of independent variables for nonlinear problems. In the current study, this method is employed to estimate discharge coefficient C_M for triangular and asymmetric labyrinth side weirs in terms of relevant dimensionless parameters. Ramamurthy et al. (2006) used this method in hydraulic engineering for the first time. As this method has received less attention in

Table 1. Range of the variables studied by Borghei et al. (2013), and used in the present study of triangular labyrinth side

weirs with one and two cycles

Cycles no.	L(cm)	p(cm)	δ(∘)	Fr_1 (–)	Q_1 (L/s)	Runs no.

1 60 – , 10, 15 – , 90, 120, 140 0.28–0.49 19.3–49.4 16 40 5, 10, 15 60, 90, 120, 140 0.17–0.67 9.8–31.8 44 30 5, 10, 15 60, 90, 120, 140 0.19–0.56 5.4–38.1 46

2 60 – , 10, 15 60, 90, 120, – 0.27–0.56 23.3–42.4 16 40 5, 10, 15 60, 90, 120, 140 0.14–0.54 8.4–33.9 46 30 5, 10, 15 60, 90, 120, 140 0.16–0.65 8.3–33.1 41

hydraulic engineering practice in the current study, this

method is employed to estimate C M for triangular and

asymmetric labyrinth side weirs.

Based on the restudy of the existing data (Borghei

et al. 2013), presented in Table. 1, the dependent vari

able C M for asymmetric labyrinth side weir to be a

function of several independent variables (L ′ /B, L ′ /L,

Fr 1 /sin(δ/2), p · sin(δ/2)/(h 1 – p), therefore: And SSE can be defined as Through using the least square method and taking derivative with respect to a ij , we may have: his equation can be reduced to Eq. (16). By using Eqs. (7) and (8): Substituting Eq. (17) in Eq. (16) yields: Using Eqs. (7) and (8) in Eq. (9) and simplifying Eq. (18), Eq. (19) is obtained:

Where t = 1, 2, 3, 4; j = 0, 1, 2, 3, 4.

Solving Eq. (19) iteratively, the constants a tj can be

obtained. Hence, the relationship among the variables

is established. Using Eqs. (9), (10), (11) and (12) in

Figs. 2a and 2b show the comparison of calculated

discharge coefficients C M for triangular labyrinth side

weirs with one and two cycles using Eqs. (20) and

(21),versus the measured data. In Figs. 2a and 2b most

of the data are within the range of ±15% with aver

age differences of 5.81% and 5.64% for triangular labyrinth side weirs with one and two cycles respec

tively. Table 2 shows a better comparison between the

accuracy of Eqs. (20) and (21) with Eqs. (1) and (2) in

predicting the discharge coefficient C_M for triangular

labyrinth side weirs.

Figure 2. Comparison of measured C_M versus calculated C_M using; (a) Eqs. (1) and (20), (b) Eqs. (2) and (21). Eq. (7) and ignoring the constants a_{tj} with negligible values, Eqs. (20) and (21) are obtained for predicting the discharge coefficient C_M of triangular labyrinth side weirs with one and two cycles respectively. 4 CONCLUSIONS In this study, the relations between the discharge coefficient C_M of triangular labyrinth side weirs with one and two cycles and their geometric and hydraulic dimensionless parameters in a rectangular open channel are presented on the basis of the multivariable nonlinear PLS method. The experimental results from 210 tests are utilized to present as De-Marchi discharge coefficient for the studied labyrinth side weirs. The Eq. (20) with Ave.

Table 2. The accuracy of Eqs. (20) and (21), compared with

the presented equations by Borghei et al. (2013) for triangular

labyrinth side weirs with one and two cycles respectively.

Triangular Labyrinth Side Weir	One cycle		Two cycles	
Accuracy	Eq. (1)	Eq. (20)	Eq. (2)	Eq. (21)
Ave. Error	5.93%	5.81%	5.51%	5.64%
NRMSE	0.3423	0.3315	0.3638	0.3685

Error = 5.81%, NRMSE = 0.3315 and Eq. (21) with

Ave. Error = 5.64%, NRMSE = 0.3685, are intro

duced to calculate the discharge coefficient of trian

gular labyrinth side weirs with one and two cycles

respectively. In comparison with the previous equa

tions, the predicted results for the triangular labyrinth side weirs discharge coefficient show an acceptable

agreement with the experimental results.

ACKNOWLEDGMENTS

This research partially was financed by the Water

Research Council, affiliated with the Ministry of

Energy of Iran.

Aydin, M. C. (2012). “CFD simulation of free-surface flow

over triangular labyrinth side weir.” Advances in Engi

neering Software, 45(1), 159–166.

Azimi, H., and Shabanlou, S. (2015). “The flow pattern in

triangular channels along the side weir for subcritical flow

regime.” Flow Meas. Instrum., 46, 170–178.

Borghei, S. M., and Parvaneh, A. (2011). “Discharge char

acteristics of a modified oblique side weir in subcritical

flow.” Flow Meas. Instrum., 22(5), 370–376.

Borghei, S. M., Nekooie, M. A., Sadeghian, H., and Jalili, M.

R. (2013). “Triangular labyrinth side weirs with one and

two cycles.” Proc. Inst. Civ. Eng., Water Manage, 166(1),

27–42.

Cheong, H.-F. (1991). “Discharge coefficient of lateral diver

sion from trapezoidal channel.” J. Irrig. Drain. Eng.,

117(4), 461–475.

Crispino, G., Cozzolino, L., Della Morte, R., and Gisonni,

C. (2015). “Supercritical low-crested bilateral weirs:

Hydraulics and design procedure.” Journal of Applied

Water Engineering and Research, 3(1), 35–42.

Emiroglu, M. E., Kisi, O., and Bilhan, O. (2010). “Predicting

discharge capacity of triangular labyrinth side weir located

on a straight channel by using an adaptive neuro-fuzzy

technique.” Advances in Engineering Software, 41(2),

154–160. Emiroglu, M. E., Aydin, M. C., and Kaya, N. (2014). “Discharge characteristics of a trapezoidal labyrinth side weir with one and two cycles in subcritical flow.” J. Irrig. Drain. Eng., 140(5), 1-13. Fritz, H. M., and Hager, W. H. (1998). “Hydraulics of embankment weirs.” J. Hydr. Eng., 124(9), 963–971. Gonzalez, J., Rojas, H., Ortega, J., and Prieto, A. (2002). “A new clustering technique for function approximation.” IEEE Trans. Neural Netw., 13(1), 132–142. Hager, W. H. (1987). “Lateral outflow over side weirs.” J. Hydr. Div., 113(4), 491–503. Nezami, F., Farsadizadeh, D., and Nekooie, M. A. (2015) “Discharge coefficient for trapezoidal side weir.” Alexandria Engineering Journal, 54(3), 595–605. Parvaneh, A. (2008). “Improving efficiency of labyrinth side weirs using asymmetric geometry.” M.Sc. thesis, Dept. of Civil Engineering, Sharif Univ. of Technology, Tehran, Iran, http://repository.sharif.edu/resource/290166/. Parvaneh,A., and Borghei, S. M. (2009). “Oblique side weir.” Proc. 33rd IAHR-ASCE Congress, 9-14 August. University of British Columbia, Vancouver, Canada. Press, No. 11310, pp. 1046–1053. Parvaneh,A., Borghei, S. M., and Jalili, M. R. (2010). “Determination of discharge coefficient for oblique side weirs using the Partial Least Square method.” Proc. ISHPF 2010 Congress, 13-15 September. Nanjing Hydraulic Research Institute, Nanjing, China. Press, Volume 2, pp. 171–177. Parvaneh, A., Borghei, S. M., and Jalili Ghazizadeh, M. R. (2011). “Discussion of discharge capacity of labyrinth side weir located on a straight channel.” Journal of Irrigation and Drainage Engineering, 137(11), 743–745. Parvaneh, A., Borghei, S., and Jalili Ghazizadeh, M. R. (2012). “Hydraulic performance of asymmetric labyrinth side weirs located on a straight channel.” Journal of Irrigation and Drainage Engineering, 138(8), 766–772. Parvaneh, A., Kabiri-Samani, A., and Nekooie, M. A. (2016). “Discharge coefficient of triangular and asymmetric labyrinth side weirs using the nonlinear PLS method." Journal of Irrigation and Drainage Engineering, DOI: 10.1061/(ASCE)IR.1943-4774.0001081. Parvaneh, A., Kabiri-Samani, A., and Parvaneh, M. (2016). "Discussion of prediction of discharge capacity over two-cycle labyrinth side weir using ANFIS." Journal of Irrigation and Drainage Engineering, (in press). Parvaneh, A., Yang, J., Javaheri, A., and Kabiri-Samani, A. R. (2016). "Discussion: Triangular labyrinth side weirs with one and two cycles" Proceedings of the Institution of Civil Engineers – Water Management, DOI: 10.1680/jwama.15.00032. Ramamurthy, A. S., Qu, J., and Vo, D. (2006). "Nonlinear pls method for side weir flows." J. Irrig. Drain. Eng., 132(5), 486–489. Ura, M., Kita, Y., Akiyama, J., Moriyama, H., and Kumar, J. A. (2001). "Discharge coefficient of oblique side-weirs." J. Hydrosci. Hydraul. Eng., 19(1), 85–96. Uyumaz,A. (1992). "Side weir in triangular channel." J. Irrig. Drain. Eng., 118(6), 965–970. Vatankhah, A. R. (2013). "Water surface profile along a side weir in a parabolic channel." Flow Meas. Instrum., 32, 90–95. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Discharge capacity of conventional side weirs in supercritical conditions

A. Parvaneh

Department of Civil Engineering, Sharif University of Technology, Tehran, Iran

M.R. Jalili Ghazizadeh

Department of Water Engineering, Shahid Beheshti University, Tehran, Iran

A. Kabiri-Samani

Department of Civil Engineering, Isfahan University of Technology, Isfahan, Iran

M.A. Nekooie

Department of Emergency Management, Malek Ashtar University of Technology, Tehran, Iran

ABSTRACT: Side weirs are widely used for level control and flow regulation in hydraulic engineering appli

cations such as irrigation, land drainage, and sewer systems. These hydraulic structures allow a part of the flow

to spill laterally when the surface of the flow in the main channel rises above the weir crest. The supercritical flow over a side-weir is a typical case of spatially varied flow with decreasing discharge. This study is aimed at investigating the variations of specific energy along the side-weir using experimental results. Some diagrams have been presented in terms of dimensionless ratio of diverted discharge to total discharge. These diagrams which can be used in the design of side weirs were obtained from the results of over 100 conducted tests on the supercritical flow. According to the obtained results, for the tests with upstream discharge of 45 lit/sec, the second-order relationship $Q/Q_1 = \propto X^2 + \beta X + 1$ is valid between dimensionless parameters, namely Q/Q_1 and $X = x/B$. Moreover, $q = dQ/dx$ varies linearly with respect to X . Two linear equations have been developed for tests with the discharge of 45 lit/sec to determine $\propto$, β in terms of studied dimensionless parameters.

1 INTRODUCTION

A side-weir is a hydraulic control structure widely used in irrigation and drainage networks and water treat ment plants.As well as discharge adjustment, side weir can be served for preliminary treatment and decrease of the sediment load. The other advantages of side weir lies in its capability of diverting flood flows with minimum turbulence and energy loss.

Despite of extensive studies of different types of side weirs in supercritical conditions including Emiroglu et al. (2010), Borghei and Parvaneh (2011), Parvaneh et al. (2011), Parvaneh et al. (2012), Borghei et al. (2013), Bonakdari et al. (2015), Zaji et al. (2016), and Parvaneh et al. (2016), only a limited number of studies have been carried out on the supercritical flow over rectangular side weirs.The first equation was proposed by De-Marchi (1934) based on the assumption of constant specific energy along weir length. Later, Subramanya and Awasthy (1972) applied this equation to supercritical flow and developed the following equation to determine the coefficient:

Where, C M is the discharge coefficient of side-weir

and Fr 1 is Froude number at the beginning of the weir. Equation 1 has been presented for Froude number between 2 and 4. Hager (1987) developed some equations to determine profile and discharge in the suband supercritical flows using modification of energy principle. The tests were performed on the side-weir with fixed length and limitation of Fr 1 < 2. In relation with supercritical flow, other studies including Hager (1994), Uyumaz and Musulu (1985), Granata et al. (2013) and Crispino et al. (2015) for flow over circular channels Jalili Ghazizadeh (1994), Oliveto et al. (2001), and Ghodsian (2003) can be mentioned for flow over rectangular channels. In the current research, the variations of energy along weir length has been investigated. The obtained results revealed that the amount of energy loss is notable in the supercritical flow and assumptions made in the De-Marchi’s equation are not applicable in this type of flow. 2 ENERGY AND MOMENTUM EQUATIONS IN THE ANALYSIS OF SIDE WEIRS The amount of energy at each section of side-weir can be best determined from:

Where, Z is channel height above an arbitrary datum, y

is flow depth, α is the kinetic energy correction factor,

Q is flow discharge, A is cross-section area of flow and g is the acceleration due to gravity. By defin

ing $dA/dx = B\ dy/dx$, $dH/dx = -S_f$, $dZ/dx = -S_0$

and derivation of Eq. 2 with respect to coordinate x,

water surface profile equation can be written as (Hager

1994):

Where, S_0 is channel bed slope, S_f is energy line

slope and B is width of rectangular channel. The

water surface profile can be represented by using

the momentum equation, considering an appropriate

control volume with applied forces and ignoring the

friction force (Hager 1994):

Where, β is the momentum correction factor, V is

mean velocity of flow, U is longitudinal component

of velocity of the flow. By way of comparison of equa

tions 3 and 4, it was found that through considering

$\propto = \beta$ and $U = \beta V$, the profile equations obtained

from momentum and energy equations will be the

same.

3 WATER SURFACE PROFILE WITH THE ASSUMPTION OF CONSTANT SPECIFIC ENERGY

The specific energy at each section of the channel can

be determined from:

By using the assumption of hydrostatic pressure dis

tribution in the rectangular channel, we may have:

Finally, the water surface profile can be achieved

by using equations 5 and 6 and assuming the spe

cific energy to be constant along the weir length (i.e.

dE/dx = 0) dQ/dx can be expressed by the general equation of the weirs: Where, p is the height of side-weir. The solution to this differential equation can be found by ignoring the kinetic energy correction factor and combining equations 7 and 8. In which and, C M is the discharge coefficient of side weir. DeMarchi (1934) proposed the above equations for a horizontal channel using Eq.2 and ignoring energy loss. C M is an empirical coefficient and should be determined for different types of flow conditions and side-weir geometry. 4 EXPERIMENTAL EVALUATION In order to examine the assumptions made in the current equations, supplementary tests have been carried out on the supercritical flow over side weirs. The tests were conducted in a glass channel with the length of 13 m, width of 23 cm and height of 50 cm. The side weirs made of Plexiglas were installed in the channel wall. Water surface profile at different sections was measured by piezometers installed on the channel bed and moving piezometers as well. The discharge of diverted and main flows was measured separately by calibrated side-weirs. In the current tests, Froude number is between 1 and 2. Table and Figure 1 show the variation range of the measured parameters and plan of test machine, respectively. Froude number varies within a range in which hydraulic jump can occur. Due to the importance of the current subject, the related tests to hydraulic jump and supercritical flow tests have been distinguished. 5 RESULTS ANALYSIS 5.1 Evaluation of the variations of specific energy along weir length In order to evaluate the variation of specific energy along weir length, the values of specific energy were computed at the beginning and end of weir for all conducted tests. Figure 2 provides a comparison between

Table 1. Variation range of experimental data.

Runs no.	L (cm)	p (cm)	S (%)	Q 1 (lit/s)	Fr 1
102	20, 30, 45, 60	1, 10, 19	-0.5, 0. 0.5, 1.25, 2.5	35-100	1-2

Figure 1. Plan view of the experimental set-up.

Figure 2. Comparison of specific energy at the beginning

and end of the weir.

the values of E_1 and E_2 . The mean value of decrease in

the specific energy was calculated to be 11.2% using

the following equation:

The percentage decrease in the specific energy indi

cated that the error resulting from the assumption of

constant specific energy in the supercritical flow was

larger than that in the subcritical flow.

Based on the obtained results, it was observed that

in the event of using energy equation to determine

discharge and water surface profile in the supercritical

flow, contrary to the initial assumption, C_M will not be

a constant value. This is mainly due to the considering

specific energy to be constant and ignoring energy loss

along weir length. Therefore, it can be concluded that

applying energy equation to the supercritical flow does

not lead to accurate results. Since the exact amount

of energy loss is not clear, momentum equation can

produce more accurate results. 5.2 Variations of the ratio of diverted discharge to total discharge In the event of developing a relationship that relates the measurable parameters to diverted discharge, it can be employed in the design of side weirs. For this purpose, the variations of Q_w/Q_1 with different parameters were investigated by dimensional analysis; then two relationships were derived between Q_w/Q_1 and $(L/B) \cdot (E_1 - E_2)/E_1$, $(E_1 - E_2)/E_1$ .This is illustrated in Figures 3a and 3b. 5.3 Evaluation of conducted tests with upstream discharge of 45 lit/sec In accordance with Figure 4 plotted for discharge of 45 lit/sec, the relationship between Q/Q_1 and $X = x/B$ follows a second order relationship, and accordingly $q = dQ/dx$ varies linearly with respect to x. The following expressions were obtained using regression. Q_1 is the discharge upstream of the weir, Q is discharge at each point along the channel, x is longitudinal component and B is channel width.

Figure 3. Tests results; a) linear equation, b) non-linear equation.

This was proposed for the first time by Oliveto et al. (2001). Table 2 shows the related data to Figure 4. In the third column, the water depth at 17 different points along the weir length is given. The fourth column shows the values of velocity at these points. It should be noted that, due to the low difference between

Table 2. The measured and computed parameters for a weir with $Q_1 = 45$(l/s), $L/B = 2.5$, and $S = 1.25\%$.

Point no	x	y	ν	Q	Q/Q_1	X (b)	X (L)
1	0	0.101	1.433	0.043	1.000	0.000	0.000
2	80	0.095	1.439	0.041	0.942	0.267	0.102
3	130	0.096	1.442	0.042	0.961	0.433	0.166
4	180	0.091	1.446	0.040	0.911	0.600	0.229
5	230	0.091	1.449	0.040	0.910	0.767	0.293
6	280	0.089	1.453	0.039	0.889	0.933	0.357
7	330	0.081	1.457	0.036	0.818	1.100	0.420
8	365	0.078	1.459	0.034	0.784	1.117	0.465
9	385	0.081	1.461	0.036	0.818	1.183	0.490
10	435	0.077	1.464	0.034	0.776	1.450	0.554
11	485	0.074	1.468	0.033	0.755	1.617	0.618
12	535	0.068	1.471	0.030	0.693	1.783	0.682

13	585	0.069	1.475	0.030	0.702	1.950	0.745
14	635	0.068	1.479	0.030	0.690	2.117	0.809
15	685	0.062	1.482	0.028	0.638	2.283	0.873
16	735	0.061	1.486	0.027	0.626	2.450	0.836
17	785	0.061	1.489	0.027	0.628	2.617	1.000

upstream and downstream velocity of the weir, the velocity was assumed to be linear. The discharge in the main channel is presented in the fifth column using two previous columns. And finally, the values of X (L) = x/L, X (b) = x/B, and Q/Q_1 are cited in the sixth to eights columns, respectively. The results of the design of three weirs with the same length and different channel slope are cited in Figure 5. Among the total of 102 tests, 14 tests were conducted with the discharge of 45 lit/sec and the values of ∝, β corresponding to each second-order equation were determined. Then, downstream discharge Q_1 was computed using $Q/Q_1 = \propto X^2 + \beta X + 1$ and finally (Q_w) calculated was calculated using $Q_2 - Q_1$. Afterwards, the error percentage of E was determined using Eq.16. Figure 4. Variations of Q/Q_1 with X = x/B for a weir with Q_1 = 45(l/s), L/B = 2.5, and S = 1.25%.

Figure 5. Variations of Q/Q_1 with X = x/B for a weir with

Q_1 = 45(l/s), L/B = 1.0, and S = 0, 0.5, 1.25%.

The results of 14 tests are presented in Table 3. The

mean error in the calculation of Q_w was determined to

be equal 3.16 % Eq. 17.

Where, Q_{wc} and Q_{wm} are computed and measured dis

charges, respectively, N is the number of tests and i

is the numerator of the test. The accuracy of the mea

sured values can be assessed by the Normalized Root

Mean Square Error (NRMSE) parameter, defined as:

Where, F(x) is computed value, f (x) is measured value

and f is the mean of measured values. The acceptable

Table 3. Results of 14 tests with the discharge of 45 l/s.

Run no	L(m)	p(m)	S_0 %	y_1 (m)	∝	β	R^2	Q_{wm} (m^3/s)	Q_{wc} (m^3/s)	Error%
1	0.20	0.1	0.00	0.121	0.1776	-0.3458	0.988	0.00673	0.00686	1.89
2	0.20	0.1	0.50	0.110	0.2575	-0.3469	0.930	0.00518	0.00515	0.27
3	0.20	0.1	1.25	0.100	0.2599	-0.3246	0.926	0.00467	0.00448	4.07
4	0.30	0.1	0.00	0.119	-0.0402	-0.1666	0.979	0.00980	0.00495	3.55
5	0.30	0.1	0.50	0.107	-0.0758	-0.0985	0.959	0.00842	0.00806	4.30
6	0.30	0.1	1.25	0.096	-0.1222	-0.0278	0.874	0.00730	0.00701	3.93
7	0.45	0.1	0.00	0.122	0.0207	-0.2400	0.989	0.01419	0.01446	1.89
8	0.45	0.1	0.50	0.116	0.0751	-0.2900	0.960	0.01214	0.01203	0.92
9	0.45	0.1	1.25	0.101	0.0203	-0.1819	0.930	0.01006	0.01033	2.74
10	0.45	0.1	2.50	0.091	0.0473	-0.1899	0.808	0.00790	0.00801	1.40
11	0.75	0.1	-0.50	0.120	0.0180	-0.2440	0.990	0.02156	0.02263	4.97
12	0.75	0.1	0.00	0.119	0.0179	-0.2354	0.992	0.02046	0.02145	4.83
13	0.75	0.1	1.25	0.101	0.0011	-0.1542	0.974	0.01616	0.01718	6.29
14	0.75	0.1	2.50	0.089	-0.0063	-0.1052	0.883	0.01356	0.01399	3.17

Figure 6. Comparison of measured and computed Q_w . value for NRMSE was 0.103 for all tests.The summary of the final results of 14 tests are illustrated in Figure 6. Finally, equations 19 and 20 have been developed for tests with the discharge of 45 lit/sec to determine ∝, β in

terms of dimensionless parameters given in Table 4. 6
CONCLUSION 1. Experimental evaluations indicated that energy loss for the spatially varied flow in supercritical conditions is notable and cannot be ignored, but it should be considered as a main parameter in the design of weir dimensions.

Table 4. Dimensionless parameters of 14 studied tests.

NUM	S_0 %	Fr_1	L/B	p/y_1	∝	β
1	0.00	1.13	0.67	0.083	0.1776	-0.3458
2	0.50	1.30	0.67	0.091	0.2517	-0.3469
3	0.00	1.50	0.67	0.100	0.2599	-0.3246
4	0.00	1.16	1.00	0.084	-0.0402	-0.1666
5	0.50	1.38	1.00	0.094	-0.0758	-0.0985
6	1.25	1.64	1.00	0.105	-0.1222	-0.0278
7	0.00	1.14	1.50	0.082	0.0207	-0.2400
8	0.50	1.21	1.50	0.086	0.0751	-0.2900
9	1.25	1.50	1.50	0.099	0.0203	-0.1819
10	2.50	1.73	1.50	0.110	0.0473	-0.1899
11	-0.50	1.12	2.50	0.083	0.0180	-0.2440
12	0.00	1.13	2.50	0.084	0.0179	-0.2354
13	1.25	1.44	2.50	0.099	0.0011	-0.1542
14	2.50	1.76	2.50	0.112	-0.0063	-0.1052

2. De-Marchi’s equation is not applicable to the super critical flow and the rate of diverted discharge has a direct relation with energy equation.

3. The specifications of side weir can be determined using design diverted discharge and the specifica tion of main channel.

4. For the tests with upstream discharge of 45 lit/sec, the second-order relationship $Q/Q_1 = \propto X^2 + \beta X + 1$ is valid between dimensionless parameters, namely Q/Q_1 and $X = x/B$. Therefore, $q = dQ/dx$ varies linearly with respect to X .

5. Equations 19 and 20 have been developed for tests with the discharge of 45 lit/sec to determine ∝, β in terms of dimensionless parameters given inTable 4.

ACKNOWLEDGMENT

This research was made possible by the comprehen sive technical support of the Water Research Center, affiliated with the Ministry of Energy, Their support is greatly acknowledged.

NOTATION

The following symbols are used in the present discus sion:

B = width of main channel

C_M = discharge coefficient for side weir

E = specific energy

Fr = Froude number in the channel

g = acceleration due to gravity

L = length (width) of side weir

p = weir height

Q = discharge in the main channel

Q_w = discharge over side weir S = channel slope

v= velocity in main channel

X = dimensionless longitudinal coordinate

y = flow depth in the main channel

Institution of Civil Engineers – Water Management, DOI:

10.1680/jwama.15.00032.

Subramanya, K., and Awasthy, S. C. (1972). “Spatially varied

flow over side weirs”. J. Hydr. Engrg.,ASCE, 98(1), 1–10.

Uyumaz. A., and Musulu. Y. (1985).”Flow over side weirs

in circular channels. “J.Hydr. Engrg., ASCE, 111(1),

144–160. Zaji,A. H., Bonakdari, H., Khodashenas, S. R., and Shamshirband, S. (2016). “Firefly optimization algorithm effect on support vector regression prediction improvement of a modified labyrinth side weir’s discharge coefficient.” Applied Mathematics and Computation, 274, 14–19.
Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Discharge coefficient of oblique labyrinth side weir

A. Kabiri-Samani

Department of Civil Engineering, Isfahan University of Technology, Isfahan, Iran

A. Parvaneh

Department of Civil Engineering, Sharif University of Technology, Tehran, Iran

M. A. Nekooie

Department of Emergency Management, Malek Ashtar University of Technology, Tehran, Iran

ABSTRACT: Labyrinth side weir is one type of weirs, which can be used when the length of opening is

limited. One of the advantages of labyrinth side weir is to increase the effective length of weir perpendicular

to the flow and, therefore, diverting more discharge with the same flow depth and weir geometry (opening and height). Discharge coefficient should be determined to investigate the weir performance and estimate the discharge passing over the weir. In this paper, hydraulic performance of labyrinth side weir with asymmetric geometry has been experimentally studied. The change to the geometry of ordinary labyrinth side weirs causes an increase in the effectiveness of the length of weir as being more in line with the streamlines. Thus, hydraulic behavior of this kind of labyrinth side weir with a constant opening length and different heights and angle has been investigated. The results show that this kind of weir is 17.8% more efficient than the ordinary labyrinth side weir and up to 35.3% more efficient respect to conventional rectangular side weir in a rectangular channel.

Finally, the discharge coefficient as a function of geometrical and flow variables are presented.

1 INTRODUCTION

Side weir, as a flow diverting structure in rivers and channels, is used to control discharge in the main stream. Common types of side weir are installed in the channel side, parallel to the flow direction and with a height of lower than channel height. When water level rises, some portion of the flow are deviated later ally from the weir. This performance will control the discharge and water level in the channel.

Other applications of side weirs include flood con trol and diversion in the dam reservoirs, flow division and protection against floods in channels and rivers. Moreover, side weir is characterized as one the major hydraulic protective structure in the water conveyance systems. A schematic view of the hydraulic of side weir is illustrated in Figure 1.

The study of side weir originated in the twenty cen tury. However, the studies conducted by De-Marchi (1934) were the base of other studies. He assumed that specific energy is constant along weir length. Most of researches including current paper have been fol lowed more closely on the basis of this assumption. On the other hand, some researchers believe that specific energy is not constant along weir length and as such proposed some methods for flow analysis in the side weirs (El-Khashab and Smith 1976). The main objec tive of conducted studies was to derive a relation for discharge coefficient and finally to estimate passing discharge from weir. Figure 1. A schematic view of side weir, a) section, b) plan. De-Marchi (1934) by assuming constant energy across the weir was among the first to introduce the discharge equation over side weir. Therefore, by taking $S_0 - S_f = 0$, where S_0 is the channel slope and S_f is the energy slope, the differential equation of flow becomes (Chow 1959);

Where y is the water depth, x is the flow direction, Q is the discharge, g is the gravitational acceleration, B is the channel width, and dQ is the discharge variation in

the main channel in flow direction. Also the discharge

outflow for an element over the weir can be written as;

Where w is the weir height and C M is known as the

De-Marchi coefficient and is a function of geometrical

and flow variables. Due to the assumption that specific

energy is constant in the channel, flow rate at sections

can be computed as follows:

Substituting Eq.1 and 2 in 3 yields:

Performing integration of Eq. 4 and assuming that C M

is independent of x, we obtain

In which �(y, E, w) is;

Thus, the relation between the length of the weir (L)

and other hydraulic variables of the flow would be:

Table 1. The value of C M according to some researchers.

Subramanya (1972) 0.864(1 + Fr 2 1 − Fr 2 1 /2) 0.5
0.02–0.85 0.20–0.96 –

Hager (1987) 0.485((2 + Fr 2 1)/(2 + 3Fr 2 1)) 0.5
0.00–0.87 – 0–20

Cheong (1991) 0.30 − 0.14(Fr 1) 2 0.28–0.78 0.42–0.85 –

Singh et al. (1994) 0.33 − 0.18(Fr 1) + 0.49(w/y 1)
0.23–0.43 0.42–0.85 –

Borghei et al. (1999) 0.7 − 0.48(Fr 1) + 0.3(w/y 1) +
0.06(L/b) 0.10–0.90 – 1–19 Where, 2 and 1 are upstream and
downstream amount of equation 6, respectively. Considering
Q 1 and Q 2 as upstream and downstream discharges,
respectively, the outlet discharge from weir (Q W) is
described by equation 8. After proposing De-Marchi
equation, a significant amount of studies have been
conducted to determine C M . Some of empirical elations for
discharge coefficient of normal labyrinth side weir are

listed in table 1. Yet not too many studies have been performed on the labyrinth side weirs. Ura et al. (2001) carried out analytical and experimental studies on the side weirs to minimize disadvantages of side weir. They believed that normal side weirs have some disadvantages such as flow segregation and decrease of discharge coefficient as a result of increase of Froude number in the channel. Therefore, they employed a labyrinth side weir and presented equation 9 for discharge coefficient in terms of upstream Froude number (Fr_1) and angle of diversion (θ) (Figure 2). Taheri et al. (2005) carried out some studies on the oblique side weir through fixing the chord length and vertical angle and changing the angle θ. In other words, the opening and effective length were fixed and variable, respectively. They conducted several tests on the different dimension and hydraulic of weir in two cases (Figure 2). In the first case, water only exits from one side of the weir ($L' = L\sin\theta$), whereas, in the second case, water can exit from both sides of the weir ($L' + L'' = L\sin\theta + L\cos\theta$). They proposed equation 10 and 11 for the first and second case, respectively.

Figure 2. Plan view of oblique side weir.

Over the course of past few years, labyrinth side weirs have been among the most popular structures studied by active researchers and engineers in the field of irrigation engineering. A labyrinth weir is a zigzag plane shaped weir and provides a longer total effective length for a given overall spillway width. The purposes of this type of weirs are to increase outflow discharge and increase water storage through increase in the crest height. The most important parameters in this weir include number and angle of labyrinths, and height, thickness and shape of crest. When the opening length is limited and a specified discharge should be released from the system, labyrinth side weir can be used through combination of side weir and labyrinth weir.

Figure 3 shows the plan view of labyrinth side weir.

According to the literature review, it has been real

ized that although there are many studies on the

hydraulic discharge and characteristics of triangu

lar labyrinth weirs, including Parvaneh and Borghei

(2009), Emiroglu et al. (2010), Parvaneh et al. (2010),

Parvaneh et al. (2011), Borghei et al. (2013), and

Nezami et al. (2015) only Borghei and Parvaneh

(2011), Parvaneh et al. (2012), Bonakdari et al. (2015),

Zaji et al. (2016), and Parvaneh et al. (2016) studied the

performance of asymmetric labyrinth side weirs. The

main difference of triangular and asymmetric labyrinth

side weirs lie in the symmetry of the weir with regard

to crown. Borghei et al. (2013) developed equation 12

and 13 for the side weir with single labyrinth through

analysis of the results of two side weirs using SPSS

Software and non-linear partial least square method. Figure 3. Plan view of labyrinth side weirs; a) triangular, b) asymmetric. He also showed that the passing discharge from labyrinth side weir increased 20 percent compared to that from normal side weir. Because only a few studies have been conducted on the labyrinth side weirs and also the efficiency of labyrinth side weir is by far greater than normal side weir, it is anticipated that labyrinth side weir will be the focus of a significant amount of analytical and experimental studies in the coming years. This research is aimed at determination of discharge coefficient of labyrinth side weir with new geometry by using experimental tests. This change caused increase in both orthogonality length and effective length of side weir and consequently discharge coefficient increased and side weir performance improved. 2 THE PLAN VIEW OF LABYRINTH

SIDE WEIRS In this paper, triangular labyrinth side weir (Figure 3.a) proposed by Emiroglu et al. (2010), and Borghei et al. (2013) has been converted into asymmetric labyrinth

Table 2. Range of the variables studied by Parvaneh (2008) and Parvaneh et al. (2012)

θ' (°)	L (m)	w (cm)	w/y_1 (-)	Q (m^3/s)	Fr_1 (-)	Number of runs
30	0.3	5, 7.5, 10, 15	0.46-0.83	0.019-0.030	0.19-0.96	40
	0.4	5, 7.5, 10, 15				
45	0.3	5, 7.5, 10, 15	0.46-0.83	0.019-0.030	0.19-0.96	55
	0.4	5, 7.5, 10, 15				
	0.6	5, -, 10, 15				
60	0.3	5, 7.5, 10, 15	0.46-0.83	0.019-0.030	0.19-0.96	50
	0.4	5, -, 10, 15				
	0.6	5, -, 10, 15				
70	0.3	5, 7.5, 10, 15	0.46-0.83	0.019-0.030	0.19-0.96	55
	0.4	5, 7.5, 10, 15				
	0.6	5, -, 10, 15				

side weir (Figure 3.b). In other words, while side A was fixed, side B was approached to the channel. As a result, this led to increase in the orthogonality of streamlines and also effective length equal to length of new side C. Accordingly, it is expected that discharge coefficient increases and this in turn, causes a further increase and decrease in discharge and water nappe height over crest, respectively.

3 EXPERIMENTAL RESEARCH

The experimental model involves an inclined rectan gular section flume with the width 40 cm and effective length of 11 m. The channel wall is made of glass and has a high of 66 cm. Also, the channel bottom is color coated metal. Water circulation system is a closed sys tem in which water is conducted from main reservoir to glass flume using pump. Prior to reaching to the side weir, some portion of the flow in the glass flume deviate laterally and discharges into the lateral reservoir. After measuring the amount of lateral discharge Q_W by V-Notch weir of the reservoir, it flows into the main reservoir. The remainder flow in the flume discharges into an end reservoir at the end of flume and after measuring the amount of discharge (Q_2), it flows into the main reservoir. The side weir used in the tests were made of Plexiglass. Altogether, 200 tests were conducted. These tests were performed for opening length of 30, 40 and 60 cm, height of 5, 7.5, 10 and 15 cm and angle of 20°, 30°, 45° and 60° of a modified single labyrinth. Table 2 presents the tests variables.

4 RESULTS AND DISCUSSION

First, the variations of specific energy along the length of side weir were studied. The amount of specific energy at upstream and downstream of weir has been compared in Figure 4. The change percentage of specific energy in the conducted tests was 1.29% which indicates minor change in the specific energy. Therefore, it is possible to assume that specific energy is constant along the weir length. Figure 4. Comparison between E_1 and E_2. Developing a relation for C_M on the basis of only one dimensionless parameter has a low level of accuracy. Hence, SPSS Software has been utilized to consider simultaneous effects of parameters on C_M. In this software, independent and dependent variables are determined in such a manner to achieve maximum regression coefficient (R^2). For the sake of deriving an appropriate relation for C_M, different relations were introduced to the software. After running the software, the constant coefficients and regression coefficient are computed in the output. Among these relations, the one with higher regression coefficient is taken as an appropriate relation. To ensure that the relation has adequate accuracy, in addition to (R^2), normalized root mean square error (NRMSE) should be determined. Lower NRMSE corresponds to lower data scattering and higher accuracy of the relation. NRMSE is defined as follows:

Figure 5. Observed C_M values versus measured C_M values

from equation 15

Figure 6. Comparison of measured and computed Q_W.

Where, F(x) is calculated value, f (x) is measured value

and f is the mean of measured values. Thus, the fol

lowing nonlinear relation is obtained for labyrinth side

weir:

In which L' is effective weir length. These results have

been illustrated in Figure 5 in terms of calculated and

observed coefficients. It is apparent that the results are

mostly within the range of ±7% which indicates good

agreement with the experimental results.

To study the effect of variations of C_M, it is required

to determine Q_W using proposed C_M and then compare

with measured values. For this purpose, through using

Eq. 7 and substituting y_1, y_2, w in 1, 2 and also

L and C_M in Eq.15, we obtain a system of equations with two unknowns in terms of E_1, E_2. Solving this system of equations gives expressions for (E_1) calculated and (E_2) calculated. By using these parameters and Eq. 3, the amount of (Q_1) calculated and (Q_2) calculated will be determined. Finally, (Q_W) calculated would be the difference these two parameters. Figure 6 provides a comparison between measured and computed discharge Q_W. 5 CONCLUSION According to the literature review, it has been realized that although there are many studies on the side weirs, only a few studies have been carried out on the asymmetric labyrinth side weir. Due to the lack of sufficient information about the performance of this type of side weirs, further development seems necessary through ongoing research. In this study, a labyrinth side weir with different geometry has been proposed. This change in the normal oblique side weir caused increase in both orthogonality length and effective length of side weir and consequently the discharge coefficient increased and side weir performance improved. Then through performing dimensional analysis, appropriate linear and nonlinear relations for discharge coefficient were obtained. The discharge coefficients and discharge were within the range of ±5% and ±7%, respectively. Moreover, the results revealed that the deviated discharges by modified oblique side weir have increased about 17.8% and 35.3% for labyrinth and normal side weir, respectively.

ACKNOWLEDGMENT This research was made possible by the comprehensive financial and technical support of the Water Research Center, affiliated with the Ministry of Energy. Their support is greatly acknowledged.

6 NOTATION The following symbols are used in this paper: b width of channel C_M side weir discharge coefficient or De-Marchi coefficient E specific energy Fr Froude number in the main channel g acceleration due to gravity L length (width) of side weir Q discharge in the main channel Q_W discharge over side weir w height of weir crest x distance alongside weir y flow depth in the main channel θ triangular labyrinth side weir included angle θ′ asymmetric labyrinth side weir included angle

Bonakdari, H., Zaji,A. H., Shamshirband, S., Hashim, R., and

Petkovic, D. (2015). “Sensitivity analysis of the discharge

coefficient of a modified triangular side weir by adap

tive neuro-fuzzy methodology.” Flow Meas. Instrum., 73,

74–81.

Borghei, S. M., Jalili, M. R., and Ghodsian, M. (1999).

“Discharge coefficient for sharp crested side weirs in

subscritical flow.” J. Hydr. Engrg., ASCE, 125(10), 1051-

1056.

Borghei, S. M., and Parvaneh, A. (2011). “Discharge char

acteristics of a modified oblique side weir in subcritical

flow.” Flow Meas. Instrum., 22(5), 370–376.

Borghei, S. M., Nekooie, M. A., Sadeghian, H., and Jalili,
M.

R. (2013). “Triangular labyrinth side weirs with one and

two cycles.” Proc. Inst. Civ. Eng., Water Manage, 166(1),

27–42.

Chow, V. T. (1959). “Open channel hydraulics” New York,

McGraw-Hill Book Company, Inc.

Cheong, H. F. (1991). “Discharge coefficient of lateral
diver

sion from trapezoidal channel”. J. Irrig. and Drain. Engrg.,

ASCE, 117(4), 461–475.

De-Marchi, G. (1934). “Sagio di teoria di fuzionamente degli

stramazzi laterali.” L’Elettrica, Millano, Italy, 11(11),

849–860.

El-Khashab, A., and Smith, K. V. H. (1976). “Experimental

investigation of flow over side weirs.” J. Hydr. Engrg.,

ASCE, 102(9), 1255–1268.

Emiroglu, M. E., Kaya, N., and Agaccioglu, H. (2010). “Dis

charge capacity of labyrinth side weir located on a straight channel." J. Irrig. Drain. Eng., 136(1), 37–46.

Emiroglu, M. E., Kisi, O., and Bilhan, O. (2010). "Predicting discharge capacity of triangular labyrinth side weir located on a straight channel by using an adaptive neuro-fuzzy technique." Advances in Engineering Software, 41(2), 154–160.

Hager, W. H. (1987). "Lateral outflow over side weirs".ASCE Journal of Hydraulic Engineering, 113(4), 491–503.

Nezami, F., Farsadizadeh, D., and Nekooie, M. A. (2015) "Discharge coefficient for trapezoidal side weir." Alexandria Engineering Journal, 54(3), 595–605.

Parvaneh, A. (2008). "Improving efficiency of labyrinth side weirs using asymmetric geometry." M. Sc. thesis, Dept. of Civil Engineering, Sharif Univ. of Technology, Tehran, Iran, http://repository.sharif.edu/resource/290166/.

Parvaneh,A., and Borghei, S. M. (2009). "Oblique side weir." Proc. 33rd IAHR-ASCE Congress, 9–14 August. University of British Columbia, Vancouver, Canada. Press, No.11310, pp. 1046–1053. Parvaneh,A., Borghei, S. M., and Jalili, M. R. (2010). "Determination of discharge coefficient for oblique side weirs using the Partial Least Square method." Proc. ISHPF 2010 Congress, 13–15 September. Nanjing Hydraulic Research Institute, Nanjing, China. Press, Volume 2, pp. 171–177. Parvaneh, A., Borghei, S. M., and Jalili Ghazizadeh, M. R. (2011). "Discussion of discharge capacity of labyrinth side weir located on a straight channel." Journal of Irrigation and Drainage Engineering, 137(11), 743–745. Parvaneh, A., Borghei, S., and Jalili Ghazizadeh, M. R. (2012). "Hydraulic performance of asymmetric labyrinth side weirs located on a straight channel." Journal of Irrigation and Drainage Engineering,

138(8), 766–772. Parvaneh, A., Kabiri-Samani, A., and Nekooie, M. A. (2016). "Discharge coefficient of triangular and asymmetric labyrinth side weirs using the nonlinear PLS method." Journal of Irrigation and Drainage Engineering, DOI: 10.1061/(ASCE)IR.1943-4774.0001081. Parvaneh, A., Kabiri-Samani, A., and Parvaneh, M. (2016). "Discussion of prediction of discharge capacity over two-cycle labyrinth side weir using ANFIS." Journal of Irrigation and Drainage Engineering, (in press). Parvaneh, A., Yang, J., Javaheri, A., and Kabiri-Samani, A. R. (2016). "Discussion: Triangular labyrinth side weirs with one and two cycles" Proceedings of the Institution of Civil Engineers – Water Management, DOI: 10.1680/jwama.15.00032. Subramanya, K., and Awasthy, S. C. (1972). "Spatially varied flow over side weirs". J. Hydr. Engrg.,ASCE, 98(1), 1–10. Singh, R., Manivannan, D., and Satyanarayana, T. (1994). "Discharge coefficient of rectangular weirs." J. Irrig. and Drain. Engrg., ASCE, 120(4), 814–819. Taheri, N., and Borghei, S. M. (2005). "Experimental determination of discharge coefficient of oblique side weir", 2nd National Congress of Civil, Iran University of Science and Technology, Iran. Ura, M., Kita,Y.,Akiyama, J., Moriyama, H., and Kumar Jha, A. (2001). "Discharge coefficient of oblique side weirs", J. Hydroscience and Hydraulic Engrg., JSCE, 19(1), 85–96. Zaji,A. H., Bonakdari, H., Khodashenas, S. R., and Shamshirband, S. (2016). "Firefly optimization algorithm effect on support vector regression prediction improvement of a modified labyrinth side weir's discharge coefficient." Applied Mathematics and Computation, 274, 14–19.
Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) , ISBN 978-1-138-02977-4

Reconstruction of a stage-discharge relation for a damaged weir on the Cavaillon river, Haïti

O. Carlier d'Odeigne, O. Roelandts, T. Verschoore, Y. Zech & S. Soares-Frazão

Institute of Mechanics, Materials and Civil Engineering, Université Catholique de Louvain,

Louvain-la-Neuve, Belgium

ABSTRACT

Haiti is probably one of the most exposed countries

to dramatic floods and inundations due, among other,

to intense deforestation. Haiti is also extremely poor in hydrologic and hydrographic data, as this is not considered as a priority in a context of frequent dis asters. The ambition of the program sustained by the Belgian Cooperation Administration (ARES-CCD) is to create, through an exemplary watershed and river reach along the Cavaillon River, a simple and repeat able methodology for enhancing hydrologic data and designing flood protection procedures.

One of the required data is the discharge of the river. As a weir, located near the village Dory, represents the upstream limit of the studied reach, we have the opportunity to evaluate the Cavaillon discharge from a continuous water level measurement located 50 m upriver. Unfortunately this infrastructure is in very bad condition as time and recurrent floods have done their work and considerably degraded the initial weir profile (Figure 1).

The aim of the presented work is to build a stage discharge relation, considering that no data are avail able about the design and the construction of this weir and that the damages have the consequence that it is certainly not working in standard conditions. In situ survey, laboratory scale model and numerical mod elling using the free Open Foam software are used to address the problem from complementary approaches.

To validate the numerical approach, a flow over a scaled model, 4 cm large and 7.5 cm height (crest level) has been studied. Laboratory measurements have been conducted by image analyses, allowing similar representations of the experimental observations and of the results obtained with the numerical model (Figure 2).

A two-fluid SPH model for landslides

A. Ghaïtanellis, D. Violeau, A. Leroy & A. Joly

EDF/LNHE & Saint-Venant Laboratory for Hydraulics, France

M. Ferrand

EDF/MFEE, France

ABSTRACT: A multi-fluid weakly compressible Smoothed Particle Hydrodynamics (SPH) method is used to model non-cohesive sediment transport. The present SPH model is based on Hu and Adam's (2006) multi-fluid formulation and Unified Semi-Analytical Wall (USAW) boundary conditions (Ferrand et al., 2013). It is then able to handle accurately density and viscosity discontinuities at the interface of two immiscible fluids, guaranteeing the continuity of velocity and shear-stress across the interface. On the other hand, the USAW boundary conditions ensure an accurate pressure and shear-stress treatment at the wall even for complex boundary geometries. In addition, a rheological model is used in order to take into account the non-Newtonian behavior of the sediment.

The present work aims at modeling a submarine landslide. The collapsing mass of non-cohesive sediment (e.g. sand) is then modeled as a shear-thinning fluid using

a Bingham rheological law. The proposed formulation

as applied to Assier-Rzadkiewicz's landslide case (Assier-Rzadkiewicz et al., 1997). Results are compared with

experimental data as well as numerical results (Capone et al., 2010) and a good agreement is obtained.

1 INTRODUCTION

As common natural phenomena, landslides are a major concern for many industrial activities and environmen tal purposes, including hydroelectric energy genera tion and risk analysis. Indeed, landslides can cause tsunamis that are likely to overtop the dam, leading to disasters such as the Vajont Dam accident in 1959 in Italy. Numerical simulation of landslides makes it possible to improve the prediction of their potential consequences, and therefore to increase the safety of such hydraulic structures. Smoothed Particle Hydro dynamics (SPH) is a Lagrangian mesh-free method for numerical simulation of hydrodynamics problems. The particle-based nature of SPH makes it particu larly adapted for simulation of flows involving highly deformed interfaces. In this paper, a multi-fluid weakly compressible SPH (WCSPH) model, combined with shear-thinning rheological law is proposed in order to simulate landslides.

The current multi-fluid model is based on Hu and Adams (2006) formulation that handles accurately density and viscosity discontinuities at the interface

of two immiscible fluids, and guarantees the continu

ity of velocity and shear-stress across the interface.

However, the original Hu and Adams approach cannot

be used to simulate free-surface flows. To circumvent

this drawback, we adapted the continuity equation pro

posed by Vila (1999) to the multi-fluid framework.

The new model is then able to handle free-surface flow but is also more stable (see Vila, 1999). The boundary conditions are imposed using the Unified Semi-Analytical Wall Boundary (USAW) conditions model that has proved its efficiency in simulating flows that both require an accurate pressure and shear stress treatment at the wall, and present complex boundary geometries (Ferrand et al., 2013). A twofluid Poiseuille flow and an air-water dam-break are presented to illustrate the capability of the present multi-fluid model. To approximate the rheological behavior of the sediment, we assimilate it to a viscoelastic fluid using a Bingham type law. This model is applied to a Bingham Poiseuille flow and the numerical results are compared with the analytical solution. Finally, the simulation of a schematic submarine landslide is presented and numerical results are compared with experimental data. 2 SPH MULTI-FLUID MODEL WITH USAW BOUNDARY CONDITIONS 2.1 Governing equations For weakly-compressible flows, the conservation of mass equation reads:

with ρ the density of the fluid, t the time and u the

velocity. Then, the Lagrangian form of the conserva

tion of momentum equation reads:

with p the pressure, g the gravity and τ the shear

stress tensor. For each fluid, a state equation is used

to compute the pressure from the density:

with ρ_0 the reference density of the fluid, c_0 the numer ical speed of sound, ξ the isentropic coefficient and p_{bg} the dimensionless background pressure. The same equation of state is used for every fluid, with suitable values of ρ_0, c_0 and ξ. For air-water simulations, ξ is usually chosen at 7 and 1.4 for water and air respectively. The physical ratio between water and air speed of sound is respected.

2.2 SPH multi-fluid model with USAW boundary conditions

The SPH method– In SPH the fluid is divided in a set of macroscopic volumes, thereafter referenced as particles, that moves at fluid velocity. The SPH particles carry physical quantities such as the mass m, the position r, the velocity u, and so on. The fluid governing equations are discretized using a discrete interpolation of fields and differential operators. For a particle a, the contribution of one neighbor b to the SPH interpolation functions is weighted by a kernel function w that only depends on interparticle distance $r_{ab} = |r_a - r_b|$.

USAW boundary conditions – To impose boundary conditions with this approach, solid walls are meshed using a set of boundary elements (S) being segments in two dimensions and triangles in three dimensions. Vertex SPH particles (V) are placed at the vertices of the mesh. They are a part of the fluid but they move at wall velocity. As illustrated in figure 1, their vol ume V is calculated as a fraction θ of a reference

volume denoted V , i.e. $V = \theta V$. In two dimensions,

θ is defined as the angle between two connected seg

ments, divided by 2π . Thus, for vertex particles we

have $\in]0; 1[$. In order to have a general formulation,

we also define $= 1$ for free particles of fluid (F) and

$= 1/2$ for boundary elements. In three dimensions,

θ is calculated in a similar way using solid angles.

Moreover, for a particle of fluid a close to the walls,

the interpolation functions are underestimated because

of the lack of particles beyond the wall. To circumvent

this issue, we use a renormalization factor γ_a , defined

as follows:

with n the space dimension. Discrete SPH model – Regarding mass conservation, Vila (1999) gives a convenient SPH discrete form of equation (1). It is related to a kind of implicit interpolation and it dispenses with the divergence operator. This formulation naturally handles free-surface and also has good stability properties. Adapting it to the USAW and multi-fluid frameworks, we obtain: with m_a the reference mass corresponding to the reference volume: $m_a = \rho_a V_a$. Note that, in the present SPH model, the reference mass of SPH particles is constant. Hence, the variation of particles density is only due to the variation of their volume. Then, adapting the USAW discrete gradient of pressure (Ferrand et al., 2013) to multi-fluid we obtain an approximation of the pressure gradient as: where p is the pressure and $\nabla\gamma_{as}$ is the contribution of segment s to the gradient of γ_a . Note that discrete SPH differential operators contain a volumic term, related to free and vertex particles, and a boundary term related to boundary elements. Finally, to compute the viscous forces we use the formula proposed by Espanol (2003): with μ the dynamic viscosity and $e_{ab} = r_{ab} / r_{ab}$. Note that, this formula can be used for both Newtonian and non-Newtonian fluids (see Violeau, 2009).

Figure 1. (a) Sketch of a boundary with: a vertex particle $b \in V$, θ_v depends on the shape of the boundary; a segment $s \in S$ ($\theta_s = 1/2$); a fluid particle $a \in F$ ($\theta_a = 1$). (b) Sketch of a vertex particle in a right-angled corner illustrating the relation between volume V_b, the dimensionless angle θ_b and the reference volume V_b.

Time integration is done with a full explicit symplectic method that leads to the following scheme:

2.3 Validation of the multi-fluid model

The present multi-fluid model was tested on a two-fluid laminar Poiseuille flow which is particularly relevant to validate interactions between the fluids as well as boundary conditions. Indeed, the velocity profile is mainly influenced by the shear stresses at the interface and at the wall. The flow involves two fluids of different densities (ρ_1, ρ_2) and kinematic viscosities (ν_1, ν_2).

The two-dimensional flow is driven by a gravity force ρg oriented in the direction of the flow. Periodic open boundaries are used and the half-width of the channel is denoted L. In figure 2, the longitudinal velocity profile obtained is plotted in color. Results are presented in terms of dimensionless quantities:

Results are in excellent agreement with the analytical solution represented by the black dots. This demon

strates that the shear stresses at the wall and at the

interface are correctly calculated.

The model was also tested on a two-dimensional air

water schematic dam-break flow impacting a vertical Figure 2. Two-fluid Poiseuille flow – Flow in a periodic pipe with density and viscosity ratio of $\rho_1/\rho_2 = \mu_1/\mu_2 = 0.25$ ($z^+ > 0 \Leftrightarrow$ fluid 1). The horizontal velocity profile at the steady state is plotted in color. The black dots represent the analytical solution. Figure 3. Air-water dam-break – Sketch of the air-water dam-break flow problem corresponding to the experiment made by Buchner (2002). P_1, P_2 and P_3 are the pressure probes and φ is their diameter. wall. As illustrated in figure 3, the numerical setup is identical to the experiment made by Buchner (2002). Figure 4 shows a snapshot of the simulation. Numerical pressure at probes were compared with Buchner's (2002) experimental data and with Marrone's et al. (2011) numerical result. The model gives satisfactory results that are not presented in detail here (see Ghaïtanellis et al., 2009). 3 NON-NEWTONIAN RHEOLOGY 3.1 Bingham fluids The sediment is assimilated to a viscoelastic material modeled by a Bingham law. Thus, it possesses a yield stress τ_y under which no significant deformation can

Figure 4. Air-water dam-break – (a) Snapshots of the flow

at t = 1.4 s. Particles are colored with respect to dimension

less velocity magnitude: $u^+ = u/\sqrt{Hg}$, with u the velocity

magnitude and g the gravity.

occur. To model this behavior numerically, the viscos

ity of the material is tuned with respect to the local

scalar strain rate s defined as:

with s_{ij} the components of the strain rate tensor s that

reads:

To compute the viscosity, Papanastasiou's (1987) con stitutive equation is used:

where μ_0 is the viscosity of the material in highly

yielded regions, τ_y is the yield stress and m is a stress

growth exponent.

3.2 Validation of the rheological model

Papanastasiou's model was tested on a Bingham

Poiseuille flow. The flow is also driven by a gravity

force ρg and the following rheological parameters are

used:

Results are presented in terms of dimensionless quan

tities:

with L the half-width of the channel and $U_{Newtonian}$ the

maximum velocity of the flow when $\tau_y = 0$ $N \cdot m^{-2}$, i.e.

when the fluid is Newtonian.

Numerical results are compared with the analytical

solution obtained for a idealized Bingham material. Figure 5. Bingham Poiseuille flow –The horizontal velocity profile at the steady state is plotted in color. The black dots represent the analytical solution. Figure 6. Submarine landslide – Sketch of the submarine landslide problem corresponding to the experiment made by Rzadkiewicz et al. (1997). Figure 7. Submarine landslide – Snapshot of the simulation at t = 0.4s. However, as Papanastasiou is an only an approximation of Bingham formulation, we don't expect to match exactly the theoretical solution (see e.g. Beverly and Tanner, 1992). Nevertheless, in figure 5 we can see that numerical results (colored dots) are in very good agreement with the analytical solution (black dots).

Figure 8. Submarine landslide – Elevation of free-surface

at t = 0.4 s (top) and at t = 0.8 s (bottom) with $x^+ = x/L_2$ and

$z^+ = z/H$.

4 THE SUBMARINE LANDSLIDE

The multi-fluid model was finally combined with Papanastasiou's rheological law to simulate a schematic submarine landslide. This study is based on the experiment made by Rzadkiewicz et al. (1997).

A mass of sand, here assimilated to a Bingham material, is placed on an inclined surface under water. The experimental setup is illustrated in figure 6. At initial time, the mass of sand starts to collapse causing a small wave.

The Bingham rheological parameters of the sand used by Capone et al. (2010) were chosen by a trial error approach and the same values are set here.That is:

The numerical parameter m is chosen in such a way that the maximum dynamic viscosity given by the Papanastasiou's model is 500 Pa·s: $m = 0.499$ s. The same discretization as Capone et al. (2010) is used. Thus, the simulations were performed using approximately 28000 particles of fluid and 900 boundary particles, with an initial particle spacing of $\delta r = 0.015$ m. The typical computational time is four minutes for one second of physical time using C++/CUDA code on one GPU card.

Figure 7 shows a snapshot of the simulation at time $t = 0.4$ s. In figure 8 the free-surface elevation is com pared with Rzadkiewicz's (1997) experimental data and Capone's (2010) numerical results. A good agree ment with the experiment is obtained. Comparing with Capone's results, we observe only slight differences at t = 0.4 while at t = 0.8 s, the present formulation gives

Buchner, B. (2002). Green water on ship-type offshore structures. Ph. D. thesis, TU Delft, Delft University of Technology.

Capone, T., A. Panizzo, & J. J. Monaghan (2010). Sph mod elling of water waves generated by submarine landslides. Journal of Hydraulic Research 48(S1), 80–84.

Espanol, P. & M. Revenga (2003). Smoothed dissipative particle dynamics. Physical Review E 67(2), 026705.

Ferrand, M., D. Laurence, B. Rogers, D. Violeau, & C. Kassiotis (2013). Unified semi-analytical wall bound ary conditions for inviscid, laminar or turbulent flows in the meshless sph method. International Journal for Numerical Methods in Fluids 71(4), 446–472.

Ghaïtanellis, A., D. Violeau, & A. Leroy (2009). Application of the unified semi-analytical wall boundary conditions to multi-phase sph. In Proc. 4th international SPHERIC workshop, pp. 27–29. Hu, X. & N. A. Adams (2006). A multi-phase sph method for macroscopic and mesoscopic flows. Journal of Computational Physics 213(2), 844–861.
Papanastasiou, T. C. (1987). Flows of materials with yield. Journal of Rheology (1978-present) 31(5), 385–404.
Rzadkiewicz, S.A., C. Mariotti, & P. Heinrich (1997). Numerical simulation of submarine landslides and their

hydraulic effects. Journal of Waterway, Port, Coastal, and Ocean Engineering 123(4), 149–157. Vila, J. (1999). On particle weighted methods and smooth particle hydrodynamics. Mathematical models and methods in applied sciences 9(02), 161–209. Violeau, D. (2009). Dissipative forces for lagrangian models in computational fluid dynamics and application to smoothed-particle hydrodynamics. Physical Review E 80(3), 036705. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Hydraulic modelling strategies for flood mapping. Application to coastal area in central Vietnam

Vo Ngoc Duong

Faculty of Water Resource Engineering, University of Science and Technology, the University of Da Nang, Viet Nam

Philippe Gourbesville

Innovative City lab URE 005, Polytech'Nice Sophia, Nice Sophia Antipolis University, France

ABSTRACT: Flood map nowadays is seen as an indispensable tool in urbanism, flood prevention and miti gation. For this reason, establishing flood map is mighty necessary for developing the socio economy of a river catchment. In recent years, creating this kind of map based on hydraulic models has been applied and proved good efficiencies in mitigating the consequences of flood catastrophes to human at many regions on the world. However, because of the lack of observed data in large catchments as well as developing countries, applying this work for these regions becomes a huge challenge for hydrologists. The insufficiency of meteo hydrological data, coarse resolution of topography, land cover data . . .brings many difficulties to choose a suitable model or decide

a reasonable model structure for flood modeling in these catchments. This study via the flood modeling process

at downstream of Vu Gia Thu Bon catchment, a coastal region in Viet Nam central will compare the differences

between 1D model, 2D model, Quasi 2D model and 1D/2D coupling model for flood simulation. The study

also presents the uncertainties of input data such as topography, land use, rainfall, and boundary condition when

modeling flood events. These simulations are carried out on the modules of Mike by DHI software: Mike 11,

Mike 21, Mike Flood. The results might show strong and weak points of each model. These could help modelers

to get several judgments when selecting the model to build the flood map. This study is expected to give some

usefulnesses for flood modeling in the coastal part of a big catchment.

1 INTRODUCTION

The climate change is predicted to occur more severely

and more complexity. Under the impact of the variation

of weather factors, especially precipitation, extreme

flood event is expected to increase not only in inten

sity but also in frequency. It is thought to have an

great influence on all aspects of human society in the

next few years (Pachauri & Reisinger, 2007). Hence,

responding actively with these changes is an urgent

requirement today. EXCIMAP, (2007) noted that a

prerequisite for effective and efficient flood risk man

agement is the in-depth knowledge of the prevailing

hazards and risks throughout a river basin and areas of coastal flood risk. This includes information about the types of floods (river, coastal, lake and ground water), the probability of a particular flood event, the flood magnitude expressed as flood extent, water depth or flow velocity, and finally, the probable magni tude of damage (life, property, economic activity). The basic information about the flood event can be gained through flood modeling and exhibited via flood map. Therefore, flood map is an effective tool in responding proactively to flood disaster in the period of prepa ration and planning of disaster prevention as well as in the emergency response phase (Moel et al., 2009). Constructing the flood map together with taking into account the impact of climate change are seen as useful and indispensable process to respond to this natural phenomenon. It might help the local authority to have scientific evidences to suggest suitable policies and measures to reduce the impact of climate change. As above mention, the flood hazard map is an essential document for assessing the impact of a flood event to society, flood risk mitigation, flood management as well. Due to its important, up to date, many mapping methods have been developed with different theories such as hydrologic, meteorological and geomorphologic approaches representing the hazard or risk of flood in scale of a catchment (Ho et al., 2010). These methods are probably classified into four different types: Flood tracking, image processing, GIS topography combination and flood modeling. There are many pros and cons with each method. Although three first methods have good advantages with workload, there is a common weak point which concerns about their flexibility and their accuracy. It means that their produces do not take into account the effect of hydrological and hydraulic factors. Hence, they could not provide information related to stream flow such as speed or flood direction. These restrictions cause difficulties while forecasting the future scenario as well as assessing scale variability of inundation area under the impact of climate change. Conversely, the last method is realized by using a model which operates based on a mathe matical relation between input and output hydrological variables (Moel et al., 2009). The link between input and output variables are represented via different kinds of mathematical function which are able to consider on different aspects due to the viewpoint of developers, such as space, time, mathematical structure, etc. So flood mapping using the hydraulic model is expected to translate more accurately the happening of flood event, including distribution due to time and space as well as providing hydraulic information. Especially, this method allows simulating with different scenarios which help to forecast change tendency of flood map under the impact of catchment's factor variations such as the construction, land use or climate change.

Within hydraulic model, they are divided into sev eral types depending on their dimensionality, capa bilities and assumption in modelling water movement (Hunter et al., 2007; Wurbs, 1994). The cornerstone of these models is the fundamental governing equa tions of fluid dynamics—the continuity, momentum and energy equations (Anderson & Wendt, 1995). This equation is in fact known as the Navier-Stokes equa tions, which can be applied to solve complex fluid flows in the form of three dimensional (3D) hydraulic model (Bates & De Roo, 2000). However, this model is still so complicated to use for real case at this moment, so Navier-Stokes equations have been simplified into the form of St Venant equations (Nguyen, 2012), gen erally known as shallow water equations (Hernández et al., 2013) that have been applied to build one dimen sional and two dimensional hydraulic models reliable at a simplified level. With each kind of model, they have different advantages and disadvantages. In order to construct flood mapping for a region, the model selection depends on many factors, at least on the actual condition of catchment.

With the aims to choose the most suitable model for representing the flood event of Vu Gia Thu Bon catchment – a large catchment in central Vietnam, this study is realized by comparing the pros and cons of each kind of model: 1D model, Quasi 2D model, 2D model, 1D/2D coupling model. The result is also expected to supply a review for selecting model for flood simulation.

2 METHODOLOGY

2.1 Hydraulic models

This step aims to provide an overall view about which model type is suitable for flood mapping inVu GiaThu

Bon catchment. The model selection for Vu Gia Thu

Bon is based on many aspects. However, the first con

sideration is relied on the efficiency of each model

with flood modelling process in Vu Gia Thu Bon. Figure 1. Vu Gia Thu Bon catchment in central Viet Nam, and hydro meteorological network. The models comparing here are the products of Danish Hydraulic Institute. They consist of 1D model and quasi 2D model with MIKE 11, 2D model with MIKE 21HD, 1D/2D mode coupling with MIKE FLOOD. 2.2 Study area The Vu Gia – Thu Bon river system (Figure 1), which originates from the eastern side of the Truong Son mountain range and drains to the Vietnamese East Sea near the cities of Da Nang and Hoi An. It is the biggest coastal river system in the central region of Viet Nam. This system has two main rivers, the Vu Gia and the Thu Bon. The topography over this region is complex with the relatively narrow mountainous area on the upstream and the flat coastal zone at the downstream. Located at a tropical monsoon climate region with influence of the ocean to the east, rain and storm in this region behaves complicatedly. The average annual rainfall of this area is from 2,000 mm to 4,000 mm with 65% to 80% annual rainfall during the months from September to December. This region is usually suffered by two to four typhoons annually (RETA 6470, 2011; TO, 2005). Consequently, inundation related to typhoon is very serious. Due to the violence of climatological events, the fragile economic condition and the underdeveloped infrastructure, the natural disasters related to river flow deeply affect the socio-economy. The lost caused by flood and storm disaster annually in Quang Nam province was estimated average up to 6.26% of the GDP (Nguyen, 2011). 2.3 Model setup Due to catchment characteristics, the inundation frequently attacks at the downstream part of Vu Gia Thu Bon river system. Besides, the population and important economic bases concentrate merely at this area.As a result, it is not need to set up flood model for whole

Figure 2. MIKE 11 (1D) model set up for Vu Gia Thu Bon

river downstream.

catchment. Accordingly, the models are only com

pared at following areas which are around 1,780 km 2 at downstream on 10,350 km 2 in total. They are con sidered on the historical flood events occurring in the period of 10-15 November, 2007. The hydrologi cal boundary conditions used in these simulations are inherited from MIKE SHE model (Vo & Gourbesville, 2016), which was calibrated and validated for the whole catchment.

a. One dimensional modelling (1D)

The river system is represented at 1D model approach by using MIKE 11 as follows:

- River network: The model is developed on 23 big rivers and linking branches at the downstream (Figure 2)
- Cross sections:The geometry of each river branch is specified via cross section from the measurements and from the DEM.
- Boundary conditions: The upstream boundary con ditions are inherited from the MIKE SHE model and set up at 8 branches. The downstream ones are defined at the estuaries of Vu Gia and Thu Bon river branches.
- Hydrodynamic parameters: This part mainly focuses on riverbed resistance.These parameters are represented via Strickler roughness coefficient M.

b. Quasi two dimensional (Quasi 2D) modeling

In order to improve the simulating capacity of one

dimensional model, an external system is constructed

beside main river system (Figure 3) for increasing

the storage when water is over river banks. The new

network is representative for floodplain along the

river systems. The cross sections of new system are

extracted from DEM 10m of P1-08 VIE project. The

new is connected with old via links which are defined

as the form of the link channel which therefore typ

ically represents the embankment geometry between

parallel rivers. Figure 3. MIKE 11 Quasi (Quasi 2D) model set up for Vu Gia Thu Bon river downstream. Figure 4. MIKE 21 (2D) model set up for Vu Gia Thu Bon river downstream. The boundary conditions are set up as the case of MIKE 11 in the section 2.3a. The bed resistances of the system are inherited from last MIKE 11 model for main river system. c. Two dimensional (2D) modelling This approach is demonstrated by MIKE 21, 2D hydraulic model from DHI. The model is set up as the schema at the Figure 4. • Topography: The bathymetry is described via rectangular grid with 30 meter resolution. This data is converted for DEM 30m that was resized for DEM 15m supplied by LUCCI project. In order to increase river bed description, the DEM is continued adjusting by merging the surveyed cross sections. • Source and sink: 17 sources are defined to transmit the flood runoff from exterior to the modelling domain. These sources are extracted at the outlet of 9 sub catchments and 8 river branches in MIKE SHE model. • Evapotranspiration: this factor is input with November value in the result of (Vu et al., 2008).

Figure 5. MIKE FLOOD (1D/2D coupling) model set up

for Vu Gia Thu Bon river downstream.

• Precipitation: This simulation uses the rainfall data

which is redistributed spatially based on daily rain fall data from 15 rain gauge stations with the Kriging method (Vo & Gourbesville, 2014).

- Resistance: This parameter is represented via Strickler roughness coefficient M.

d. 1D/2D coupling modelling

The model is handled here from DHI group as well. This model is MIKE FLOOD which is developed on the coupling between 1D model to 2D model. The model set up is shown via Figure 5.

- 2D model – MIKE 21: The model is set up similarly as the last MIKE 21 mode in the section of 2.3c. Therefore, there are several changes in data because in this case the river flow will be responsible for MIKE 11, using of a river bed integrated topography is not necessary. The second change is in the source input. Instead of using 17 sources as the last 2D model, this model only introduces 9 sources which are representatives of 9 upstream sub catchments.
- 1D model – MIKE 11: This is benefits from the model in part of 2.3a. Each river branch in MIKE 11 connects with MIKE 21 via a 2 lateral links.

3 RESULT AND DISCUSSION

Max water levels from MIKE 11 model (1D) and Quasi MIKE 11 (Quasi 2D) model are used to construct flood hazard maps by interpolated technic in ArcGIS. This process is taken place with the topography 30 m. The

results are shown at the Figure 6. In the case of MIKE

21 (2D) and MIKE FLOOD (1D/2D coupling), the

flood hazard maps are extracted directly from model

with the same cell size of input data (30 m). The result

of these two models are shown at the Figure 7.

Relying on the hydrographs at three stations, we

recognize that there is a big difference between these

scenarios (Table 1).The water level augments of MIKE

21 structure in comparison with the others are quite Figure 6. Flooding area variation due to model structure – 1D model and Quasi 2D model. big. The hydrographs of this model are entirely separated toward MIKE 11, Quasi MIKE 11, MIKE FLOOD models at Giao Thuy and Cau Lau stations. These differences lead to a disparity in peak water level of Mike 21 model compared to the remaining models. The average number is that the MIKE 21 model peak is averagely higher than others 0.75 m at Ai Nghia, 2.73 m at Giao Thuy and especially 3.57 m at Cau Lau. These analysis demonstrate the uncertainty of MIKE 21, representative of two dimensional model in simulating flood event. These limitations might be from the topography quality (Vo & Gourbesville, 2015). Using 30 m DEM resolution here might not be enough to represent the topography at modeling area, at least at river bed area. The coarse resolution as this situation is potentially to reduce the performance of 2D algorithm. In addition, the lack of surveyed DEM that includes river bed is considered as a significant factor affecting on 2D model capacity. Although integrating the cross section in DEM helps to increase the river bed description but it is still not accurate enough. It seems

Figure 7. Flooding area variation due to model structure –

2D model and 1D/2D coupling model.

that the low resolution affects not only the intensity of

flood event but also the time factor.This issue is proved

by the late of peak water level of MIKE 21 model in comparison with others.Almost peaks of water level of MIKE 21 model are slower than the remaining from 3h to 9h. This limitation also has a significant influence on the modelling quality of hydraulic model.

The second model we note here is the MIKE 11 model. This one dimensional model is the simplest model for set up and modeling. The computation time is quite short. For running this 5 days flood event, MIKE 11 just spent less than 30 minute in comparison with more than 1 day of two dimensional models. How ever, beside these advantages, this kind of mode shows many weak points. These are mentioned at the last part and now are confirmed by MIKE 11 results. Similar to the previous 2D model, the water levels of MIKE 11 mode in this case are higher than Quasi 2D model and 1D/2D model coupling. Therefore, the intensity is not as big as MIKE 21 model, the higher only 1.85 m at

Giao Thuy, 0.86 m at Cau Lau, and 1.89 m at Ai Nghia Table 1. Variability of max water level due to model structure (m). Water level (m) Station 1D Quasi 2D 2D 1D/2D Giao Thuy 11.973 10.411 13.464 9.833 Cau Lau 6.388 5.798 9.386 5.265 Ai Nghia 12.773 11.279 12.261 10.489 (Table 1).This point could be explained by the absence of modelling the lateral runoff factor. The river bank fix will make the water lever being higher than reality in the case of overbank of river flow. The next limitation of 1D model is that it does not supply a function to present the hydrological factors interior of study area such as rainfall or evapotranspiration. These are expected to affect significantly on the results. Particularly with a large study area as this scenario (1780 km2), the role of interior factors is not able to neglect. Combining two above problems, we could figure out that, water lever in 1D model (MIKE 11) is higher than Quasi 2D and, 1D/2D coupling modes because it is impossible to describe the flow exchanged with flood plain.Therefore, although this model only counts exterior flooding causes via boundary conditions, the water lever is still lower than MIKE 21 what put in all inside and outside resources. The results in MIKE 11 are lower 1.49 m at Giao Thuy, 3 m at Cau Lau, higher 0.51 at Ai Nghia than MIKE 21 (Table 1). This distinction proves the prominence of 1D model in introducing river flow than 2D model. Additionally, the 1D model meets the difficulty in constructing flood map. Instead of providing directly the flood map as 2D model, 1D model only gives the water level along the river. Then, from level, the flood maps are established by interpolating in GIS model. Regarding Quasi 2D model (Quasi 2D), the results show that after taking a part of lateral river flow, the peak of water level is cutting down a lot in comparison with 1D model (MIKE 11). The reduction is so big, around 1.56 m at Giao Thuy, 0.59 at Cau Lau, and 1.49 at Ai Nghia (Table 1). These numbers prove that a significant water quantity was partly transformed and stored in flood plain. This scenario seems more reasonable than 1D model which merely defines the water run inside river banks. But in the other side, the recession limb of MIKE 11 quasi is higher than MIKE 11. It means that after reaching to peak flow, the water in flood plain returns to supplement for main river flow. Other characters of Quasi MIKE 11 are similar to MIKE 11 model. Finally, MIKE FLOOD model which is coupling between 1D/2D coupling models. These couplings allow the part of overbank water to be able to exchange easily with flood plain. Besides that, the 1D/2D coupling can describe more precisely the flow sources than 1D model due to 2D model. In particular, the issue seems more impressive with distributed modes as Mike from DHI where the boundary extraction and Table 2. Scale variability of inundation area due to model structure (hectare). Flood depth (m) <0.5 0.5-1.0 1.0-2.0 2.0-4.0 4.0-8.0 >=8 1D 3,720 3,909 7,430 10,330 3,741 307 Quasi 2D 3,339 3,922 6,585 6,765 1,970 261 2D 2,574 2,828 6,335 14,240 10,334 912 1D/2D 3,354 3,449 5,564 4,965 1,838 31

input are very flexible. The extracted point can be

defined easily, so it helps to simulate continuously the

flow into 1D/2D coupling model. Furthermore, not

only the outside flow sources, by coupling with 2D model, the 1D/2D coupling has the capacity to express the inside flow sources such as rainfall.

The different hydrographs, peak flows appearance time lead to the uncertainties in determining the flood area. This point is shown very clear in the Figure 6, Figure 7 and Table 2. There is an unevenness between flood maps. With the highest flood peak, the 2D model gives large flood area (Figure 7). The next serious one is 1D model when the total inundation area is 25,179.39 ha and deeply inundated area is 4,049.1 ha. These numbers are 19,505.34 ha; 2,231.91 ha and 15,848.64 ha; 1,869.79 ha with Quasi 2D model and 1D/2D coupling model respectively (Table 2).

4 CONCLUSION

The purpose of this paper is to present a viewpoint for selecting the hydraulic model in flood modeling. The study is realized in Vu Gia Thu Bon catchment, a flood prone area in central Viet Nam. By compar ing four model types: 1D model, Quasi 2D model, 2D model, 1D/2D coupling model, the study shows their advantages and disadvantages towardsVu GiaThu Bon flood.The result is also expected to become a reference for choosing hydraulic model.

The simulation shows that there is a big difference in result and computation time between the scenarios.

Due to algorithm and simple input data, the computation time of 1D mode is the shortest. Conversely, the 2D model is longer than others. However, the most important is the difference in flood propagation. The 2D model gives the biggest inundation area, contrary to 1D/2D coupling model. The significant dissimilarity between 2D simulation result and observed data in this study physically is not right. Nevertheless, this could be understood in the case of Vu Gia Thu Bon catchment where the lack of data for simulation is so critical, especially the topography data. The coarse DEM leads to the wrong in representing the topography and flood propagation.

Anderson, J. D., & Wendt, J. F. (1995). Computational fluid

dynamics (Vol. 206). Springer. Bates, P. D., & De Roo, A. P. J. (2000). A simple rasterbased model for flood inundation simulation. Journal of Hydrology, 236(1), 54–77. EXCIMAP. (2007). Handbook on good practices for flood mapping in Europe. Excimap (European exchange circle on flood mapping). Ho, L. T. K., Umitsu, M., & Yamaguchi, Y. (2010). Flood hazard mapping by satellite images and SRTM DEM in the Vu Gia – Thu Bon alluvial plain, Central Vietnam. In International Archives of the Photogrammetry, Remote Sensing and Spatial Information Science. Hunter, N. M., Bates, P. D., Horritt, M. S., & Wilson, M. D. (2007). Simple spatially-distributed models for predicting flood inundation: a review. Geomorphology, 90(3), 208– 225. Moel, H. de, Alphen, J. van, & Aerts, J. (2009). Flood maps in Europe – methods, availability and use. Natural Hazards and Earth System Science, 9(2), 289–301. Morales-Hernández, M., García-Navarro, P., Burguete, J., & Brufau, P. (2013). A conservative strategy to couple 1D and 2D models for shallow water flow simulation. Computers & Fluids, 81, 26–44. Nguyen, B. Q. (2011). Assessing impacts of climate change on disasters related to stream flow (flood and drought) in Quang Nam, Viet Nam. Nguyen, T. D. (2012).

Impact de la résolution et de la précision de la topographie sur la modélisation de la dynamique d'invasion d'une crue en plaine inondable. Pachauri, R. K., & Reisinger, A. (2007). IPCC fourth assessment report. IPCC, Geneva. RETA 6470. (2011). Investment, Managing water in Asia's river basins: Charting progress and facilitating – The Vu Gia-Thu Bon Basin. TO,T. N. (2005). Strategic plan for integrated water resources management of the Vu Gia Thu Bon Basin Organization. Vo, N. D., & Gourbesville, P. (2014). Rainfall uncertainty in distributed hydrological modelling in large catchments: an operational approach applied to the Vu Gia – Thu Bon catchment – Viet Nam. In 3rd IAHR Europe Congress, Book of Proceedings, 2014, Porto -Portugal. Vo, N. D., & Gourbesville, P. (2015). Analyzing the uncertainty of topography data for flood modelling. A case study at Vu gia-Thu bon catchment – Viet Nam. In 6th International Conference Water Resources and Sustainable Development (CIREDD2015), May 25–26, 2015, in Algiers, Algeria. Vo, N. D., & Gourbesville, P. (2016).Application of deterministic distributed hydrological model for large catchment – A case study at Vu Gia-Thu Bon catchment – Viet Nam. Journal of Hydroinformatics. Vu,V., Do, D., & Dang,T. (2008). Potential evapotranspiration estimation and its effect on hydrological model response at the Nong Son Basin, 24, 213–223. Wurbs, R. A. (1994). Computer models for water resources planning and management. DTIC Document.
Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Estimation of 1D-confluence model parameters in right-angled discordant beds' confluences using 3D numerical model

D. Đor devic´

Faculty of Civil Engineering, University of Belgrade, Belgrade, Serbia

I. Stojnic´

(CERIS) – Instituto Superior Técnico – TU Lisbon, Portugal

LCH – EPFL, Lausanne, Switzerland

ABSTRACT: Parameters of 1D confluence models are originally defined for concordant beds' (CB's) con

fluences. This paper aims at estimating these parameters for discordant beds' confluences using numerical simulation results of 3D flow. A 3D finite-volume based model SSIIM2 that was successfully validated in CB's confluences is applied in this study, too. Confluences with the low, moderate and maximal observed bed elevation discordance ratio values are analysed for three characteristic hydrological scenarios: dominance of the tributary flow, equal contributions of the combining flows and dominance of the main-river flow. It is shown that: 1) the mean flow angle $\bar{\delta}$ approaches junction angle α with the increase in bed elevation discordance, especially when tributary flow dominates, 2) the value of Hager's correction coefficient σ is not constant and 3) the contribution of the tributary flow to the 1D momentum equation is under predicted when either parameter $\bar{\delta}$ or σ is used for its estimation.

1 INTRODUCTION

Extensive bathymetric surveys in river confluences during mid-1980s revealed that there was a difference in bed elevations between the tributary and main channels in the majority of surveyed confluences (Kennedy, 1984). The difference was created through a combined effect of the deposition of coarse sediment particles that had been arriving from upstream channel and deepening of the scour hole at the entrance to the post confluence channel due to enhanced turbulence caused by collision of the combining flows (Con-stantinescu et al., 2011). The presence of the bed step at the trib utary entrance to the confluence affects momentum transfer from the tributary to the main-river. A proper estimation of this influence is of crucial importance for an accurate prediction (calculation) of upstream water levels (flow depths) in 1D flow modelling of dendritic river networks and lengthy river reaches. However, existing 1D-confluence models that were intended for the treatment of a confluence as an internal bound ary condition in such analyses had been developed for the concordant beds' case, i.e. for the case when bed elevations of the combining channels are equal. The early models of Taylor (1944), Weber and Greated (1965) and Lin and Song (1979) did not take into account the fact that the tributary flow deflected from the junction angle as it entered the main-river. Those proposed after 1980 (Hager, 1987, 1989; Ramamurthy et al., 1988; Gurram et al., 1997; Hsu et al., 1998a, b and Gurram & Karki, 2000) encountered the tributary flow deflection in calculating its contribution to the momentum equation for the direction of the main-river flow either by introducing the correction coefficient σ for the junction angle α (Hager, 1987, 1989 and Gurram et al., 1997), or by observing pressure difference between the opposite tributary walls near the confluence (Ramamurthy et al., 1988) or, by introducing mean cross-sectional value of the flow deflection angle in the downstream section of the tributary channel (Hsu et al., 1998a, b). All these models resulted from the combination of theoretical analysis and experiments. Experiments were used to estimate and recommend values of key parameters and/or variables that were necessary for the proper inclusion of the momentum transfer from the tributary to the mainchannel. However, to draw general conclusions about model parameters extensive laboratory experiments or field measurements are needed. Since the preparation and performance of laboratory experiments might be costly, Đorđević (2014) considered a possibility of using a 3D numerical model as a substitute to a physical model in studying the confluence hydrodynamics and in estimating parameters of 1D-confluence models. The study undoubtedly confirmed that a 3D finite-volume based model SSIIM2 was a reliable predictive tool in concordant beds'confluences. Moreover it confirmed once again the dependence of Hager's parameter σ on the discharge ratio $D_R = Q_{MR}/Q_d$ (where Q_{MR} is the discharge in the main-river upstream

Figure 1. a) Planform of Shumate's (1998) laboratory confluence, b) definition sketch for the flow angle δ, c) concordant bed's confluence, d) discordant beds'confluence.

of the confluence, and Q_d is the total downstream discharge) in the 90° confluence – a feature that had not been fully discussed and underlined in original papers of Best and Reid (1987) and Hager (1989).

This paper continues the line of Đorđević's previous study (2014), by analysing the effect of bed elevation discordance on the values of parameters from 1D concordant beds' confluence models of Hager, Gurram et al. and Hsu et al. One can conclude from Best and Reid's and Hager's studies that there is a dual dependence of the junction angle correction coefficient σ on the junction angle α, and the discharge ratio D_R. However, it is intuitively obvious that the elevated channel bed of the tributary with the backward facing step at its downstream end results in increased momentum of the tributary flow and thus in its ability to keep its orig inal direction. This paper will show how the extent of the bed elevation discordance affects the average flow deflection angle of the tributary flow on the horizontal plane ($\bar{\delta}$) and its correction coefficient σ. Additionally, the paper will attempt to evaluate the discrepancy between values of the component of the tributary force of inertia that acts in the main river direction (I_{Tx}) obtained from different 1D models and that calculated by integration of the corresponding component of the momentum flux over the downstream tributary cross-section.

To keep up with the previous study, a layout from Shumate's 90° concordant beds' confluence of equal width laboratory canals with rectangular cross-section and horizontal beds is also used in this analysis (Fig. 1a). In addition to the original confluence layout ($\Delta z_T/h_d = 0.00$), three different hypothetical layouts with the following values of the bed elevation discordance ratio $\Delta z_T/h_d = \{0.10, 0.25, 0.50\}$ are analysed (where Δz_T stands for the difference in bed elevations between the tributary and main canals and h_d for the flow depth in the main canal in the confluence, Figs 1c, d). Values of the average δ-angle ($\bar{\delta}$) and the component I_{Tx} of the force of inertia in the downstream tributary cross-section are deduced from the numeri cal simulation results for the three different D_R values: D_R = {0.250, 0.583, 0.750}. Results are obtained with the 3D finite-volume based model SSIM2 that was previously successfully validated against the data from laboratory (Biron et al., 1996) and field (Đorđević, 2010) discordant beds' confluences. Since 1D-confluence models of Hager (1987), Gurram et al. (1997) and Hsu et al. (1998a) were recalled and briefly presented in the previous study they will not be repeated here.Thus, the paper proceeds with the description of the setup of numerical experiments (section 2) and the presentation of the numerical modelling details (section 3). Values of the characteristic parameters and variables for confluences with different extent of bed elevation discordance are compared in section 4. Additionally, equations for the best fitting curves for Hagers's σ "coefficient" are given. Finally the most important conclusions drawn from this study are summarised.

2 SETUP OF NUMERICAL EXPERIMENTS

A right-angled laboratory confluence of two straight canals with horizontal concordant beds (Shumate, 1998) is used as a starting point in this study (Figs 1a, c). Such a choice follows from the successful validation of the SSIIM2 model with the experimental data from this facility. Hypothetical, discordant beds' confluences are formed by elevating the bed of the lateral canal for the amount of Δz_T (Fig. 1d). The bed step height Δz_T is chosen such that the bed elevation discordance ratio $\Delta z_T/h_d$ does not exceed the maximal observed value in river confluences, i.e. the value of 0.50 (Biron and Lane, 2008). In addition to this value, two values that correspond to moderate ($\Delta z_T/h_d = 0.25$) and low ($\Delta z_T/h_d = 0.10$) extents of bed elevation discordance are selected for this study. To allow comparison with parameters for the concordant beds'case, numerical simulations are performed with the input data from experiments with $D_R = Q_{MR}/Q_d$ = {0.250, 0.583, 0.750}. The total, downstream discharge and the flow depth at the downstream end of the main canal were the same in all experiments ($Q_d = 0.17$ m^3/s, $h_{out} = 0.296$ m).

3 NUMERICAL MODELLING

Flow in discordant beds' confluences is simulated using 3D finite-volume based model SSIIM2 (Olsen, 2012). This model solves a set of equations which consists of the mass conservation equation, Reynoldsaveraged Naiver-Stokes equations and turbulence model equations that are used to close the system of conservation laws. The standard k–ε model is used as a turbulence model closure in this paper. Equations are solved on an unstructured multiblock space grid. The SSIIM 2 model uses SIMPLE method to couple the mass and momentum equations. Since there is no other option available, the rigid-lid approach is used to represent the free-surface. Such a treatment

Figure 2. Comparison of the measured and calcu
lated non-dimensional water surface profiles at different non-dimensional lateral distances (y/B_{PCC}) from the junc
tion-side wall in Shumate's experiment with D_R = 0.583.

of the free-surface is justified by a good agreement between numerical simulation results and measure
ments as shown in Đorđević & Biron (2008), Đorđević (2013, 2014). Validation against Shumate's experi
ments has shown that values of simulated velocities are within confidence interval which corresponds to the significance level of 0.05 (Đorđević 2014). Such a good agreement is additionally confirmed in this paper by comparison of the simulated and measured non-dimensional free-surface profiles for D_R = 0.583 (Fig. 2). It is readily noticeable that the agreement between each of the two profiles is acceptable for the concordant beds' case. Discrepancies are below 4%. Similar results are obtained for the other two D_R-values. Thus, it is reasonable to believe that the free-surface treatment by the rigid-lid approach would also result in a good estimation of pressures at the rigid-lid in discordant beds' cases i.e. good prediction of both velocity fields and the free-surface. Convec
tive terms in the momentum equations are modelled

in the same manner as in the previous study, i.e. with the second-order upwind scheme.

Boundary conditions are prescribed as follows. A constant discharge is set at each inflow boundary and a constant depth is set at the outflow boundary. Remain ing dependent variables at the outflow boundary are determined from the zero gradient condition.This con dition is also applied for ε and horizontal velocities at the free-surface, while the zero discharge condition is used to calculate the vertical velocity component. The turbulence kinetic energy at the free-surface is set to the half of its bottom value (Olsen, 2000). The treatment of solid boundaries rests on the wall-law.

The computational domain in all simulations covers full lengths of the two canals (Fig. 1a). As it was the

case in laboratory experiments, such a choice ensured Table 1. Grid size in block 2 for the four analysed confluence layouts. Case Δz T /h d Grid size Type of No. [/] in block 2 confluence 1 0.00 183×37×20 Concordant beds’ 2 0.10 183×37×18 3 0.25 183×37×15 Discordant beds’ 4 0.50 183×37×10 no influence of boundary conditions on the flow within the confluence hydrodynamic zone (i.e. zone within and downstream of the confluence where effects of the collision between and combining of the two flows are felt). The multiblock space grid has two blocks each of which is an orthogonal structured grid. The block 1 covers the main canal, whereas block 2 covers lateral, tributary canal. The size of block 1 is, therefore, the same for all confluence layouts – it has 838 cells in the stream-wise, 37 cells in the lateral and 20 cells in the vertical directions. The vertical size of block 2 reduces with an increase in the bed elevation discordance ratio, while the horizontal size remains unaltered – 183 cells in the stream-wise and 37 cells in the lateral directions. Table 1, summarises the grid size in block 2 for the four considered confluence layouts. The presented block dimensions were accepted after grid sensitivity analysis based on the value of the GCI (Đorđević, 2013). 4 RESULTS AND DISCUSSION Parameters of 1D-confluence models (i.e. the mean cross sectional flow deflection angle $\bar{\delta}$ in the model of Hsu et al. and the correction coefficient σ in models of Hager and Gurram et al.) and the component of the tributary force of inertia that acts in the direction of the mean-channel flow (I_{Tx}) are estimated from the calculated velocities u and v in the downstream tributary cross-section (Fig. 1b). Mean cross-sectional angle $\bar{\delta}$ and correction coefficient σ . The mean flow angle on the horizontal plane $\bar{\delta}$ is calculated by averaging the δ-angle over the cross-section. To do this a cross-sectional distribution of the δ-angle should be known. The distribution is found by calculating the value of this angle in each point of computational mesh in the downstream tributary cross-section from the basic trigonometric relation: $\delta = \mathrm{arc\ tg}(v/u)$. Rather than presenting these distributions for the twelve cases, variations of mean values for 38 cross-sectional verticals are given in Figure 3. It is readily noticeable that the greatest variations within the cross-section are present when the main-river flow dominates ($D_R = 0.750$, Fig. 3c) and that the least flow deflection might be expected when there is a dominance of the tributary flow ($D_R = 0.250$, Fig. 3a). For the given D_R-value, the variation reduces with the increase in the extent of

Figure 3. Effect of bed elevation discordance ratio $\Delta z_T/h_d$ on the variation of mean δ-angle values in verticals of the downstream tributary cross-section for different hydrological scenarios (D_R-values).

bed elevation discordance ($\Delta z_T/h_d$). For example, the greatest deflection from the junction angle α happens in the vertical located at $l = 0.08L_{u-d}$ in the concordant beds' case ($\Delta z_T/h_d = 0.00$) when $D_R = 0.750$. The mean flow angle in this vertical ($\bar{\delta}_v$) is only 7°, i.e. the flow deflects from the junction angle by α −

$\bar{\delta}_v = 83^\circ$. For low and moderate extents of bed elevation discordance (i.e. for $\Delta z_T \leq 0.25h_d$) there is no significant difference in comparison with the concordant beds' case (min $\bar{\delta}_v \approx \{8^\circ, 11^\circ\}$ for $\Delta z_T = \{0.10, 0.25\}$). The minimal flow angle $\bar{\delta}_v$ in this vertical increases approximately four times when $\Delta z_T/h_d$ reaches its maximal value of 0.50 (min $\bar{\delta}_v = 30^\circ$). Consequently, the mean cross-sectional flow angle $\bar{\delta}$ is increased by almost 60% (from $\approx 33^\circ$ for $\Delta z_T = 0.00$ to $\approx 53^\circ$ for $\Delta z_T = 0.50h_d$, Fig. 4a). Cross-sectional

variations are still notable for the hydrological scenario

Figure 4. Effect of bed elevation discordance ratio $\Delta z_T/h_d$ on a) mean cross-sectional flow angle $\bar{\delta}$ and b) correction coefficient σ for different hydrological scenarios (D_R-values).

with almost equal contributions of the two combining flows ($D_R = 0.583$) as long as $\Delta z_T \leq 0.25h_d$ (Fig. 3b). For $\Delta z_T = 0.50h_d$ variations across the canal width become negligible as in the case when tributary flow dominates ($D_R = 0.250$, Fig. 3a). However, the average flow deflection from the junction angle is still greater (24°) than that in the concordant beds' confluence (21°) when $D_R = 0.250$. Reduced $\bar{\delta}_v$ angle variations result in greater cross-sectional mean values: $\bar{\delta} \in [\approx 50^\circ, \approx 65^\circ]$ for $\Delta z_T \in [0.00, 0.50]h_d$ (Fig. 4a). When compared to the case with $D_R = 0.750$, a percentage increase in the $\bar{\delta}$-angle value between $\Delta z_T = 0.00$ and $\Delta z_T = 0.50h_d$ is halved. With further increase in dominance of the tributary flow ($D_R = 0.250$), the average deflection from the junction angle becomes almost constant along more than $0.50L_{u-d}$. The least flow deflection (of only 13°) happens again when $\Delta z_T = 0.50h_d$. A percentage increase in the mean cross-sectional δ-angle value between the concordant beds' confluence and the confluence with $\Delta z_T = 0.50h_d$ is halved again when compared to the case with $D_R = 0.583$, and the range of $\bar{\delta}$-angle values is reduced: $\bar{\delta} \in [\approx 66^\circ, \approx 76^\circ]$ for $\Delta z_T \in [0.00, 0.50]\,h_d$. (Fig. 4a). The correction coefficient σ changes with D_R and $\Delta z_T/h_d$ in the same manner as $\bar{\delta}$ (Fig. 4b). Such a behaviour follows from its definition: $\sigma = \bar{\delta}/\alpha$. It is

Table 2. Coefficients of the best fitting curves for the σ -coefficient ($\sigma = aD_R^3 + bD_R^2 + cD_R + d$). $D_R \in [0.250, 0.583]$ $D_R \in [0.583, 0.750]$

z_T/h_d	a	b	c	d	a	b	c	d
0.00	-1.52	1.14	-0.68	0.88	3.04	-6.83	3.96	-0.019
0.10	-2.51	1.88	-0.67	0.88	5.01	-11.28	7.00	-0.611
0.25	-2.51	1.88	-0.61	0.88	5.01	-11.28	7.06	-0.606
0.50	-0.99	0.74	-0.47	0.95	1.96	-4.42	2.54	0.361

interesting to notice that the value of the σ -coefficient is generally less than proposed constant values of Hager (8/9) and Gurram et al. (0.85), except in the confluence with $\Delta z_T = 0.50h_d$ when $D_R = 0.250$. In this case, σ approaches Gurram et al.'s value of 0.85, and exceeds corresponding value from Hagers's amended curve by 6%. Additionally, σ -value in the discordant beds' confluence with $\Delta z_T = 0.25h_d$ approaches the value from Hagers's amended curve when $D_R = 0.250$. For $D_R > 0.40$ σ -curves are always between Hager's amended and Hsu et al.'s curves. It seems that both Hager and Gurram et al. overestimated $\bar{\delta}$-angle values, because they deduced them from the point measurements with a miniature angle meter at the mid-depth where variability of the δ-angle is less pronounced than that in the bottom layers. As for comparison with Hsu et al.'s data it is should be mentioned that the tributary canal was narrower than the main-canal, in their experiments which, most probably, resulted in greater variability of the $\bar{\delta}$-angle and, thus, lower σ -coefficient values. Table 2 summarises information on best fitting curves for the four analysed confluence layouts. The best fitting curves are cubic polynomials: $\sigma = aD_R^3 + bD_R^2 + cD_R + d$. Curves are defined in the two ranges of D_R -values: $D_R \in [0.250, 0.583]$ and $D_R \in [0.583, 0.750]$.

Tributary force of inertia and its components. Components of the tributary force of inertia are calculated through the integration of corresponding momentum fluxes over the downstream tributary cross-section and the magnitude of the total force of inertia is then found from: $I_T = (I_{Tx}^2 + I_{Ty}^2 + I_{Tz}^2)^{1/2}$. In this particular case, the cross-section is a vertical plane with the normal in the y-direction (Fig. 1a). Thus, the momentum of the tributary flow is carried to the main-canal by the velocity component in the y-direction, i.e. by the v velocity. This further means that the momentum flux through the elemental surface dA_y is described for each coordinate direction only with one term from every momentum conservation equation. The flux in the direction of the main-canal axis is described by the term $\rho uv dA_y$ from the momentum equation in x direction, that for the lateral direction is described by the term ρvvdA y from the equation for the y-direction,

while the term ρwvdA y from the equation for the z

direction describes the flux through dA y in the vertical,

z-direction. The latter two components are relevant

only for the calculation of the total force of inertia

(I T), while the one in the x-direction is of interest for

the 1D confluence flow modelling. Figure 5. a) Variation of the I Tx magnitude with D R and �z T /h d , b) contribution of the I Tx magnitude to the magnitude of the total force of inertia I T , c) contribution of the I Ty magnitude to the magnitude of the total force of inertia I T . A variation of the I Tx magnitude with D R and�z T /h d is presented in Figure 5a. One can observe from this figure that the I Tx magnitude decreases with the increase in dominance of the main-river flow regardless the bed step height at the tributary entrance to the confluence and that it increases with the increasing extent of the bed elevation discordance between the two canals for the given D R -value. However, such an observation may be misleading in deriving conclusions about the effect of �z T /h d on the contribution

Figure 6. Comparison of the force of inertia I Tx to its esti

mates based on the mean cross-sectional flow angle δ¯ (Hsu

et al.) and the proposed σ correction coefficient value of 8/9

(Hager).

of the tributary force of inertia to the 1D momentum

equation. Therefore, a contribution of the I Tx mag

nitude to the magnitude of the total force of inertia

I T is presented in Figure 5b. This figure shows what

was intuitively expected. Firstly, that the increased

deflection of the tributary flow, when the main-canal flow dominates (D R = 0.750, Figs 2c and 3a), results in greater contribution of the I Tx component to the total force of inertia, and, thus, greater contribution of the tributary flow to the 1D momentum equation. Secondly, the momentum in the direction of the tribu tary canal, i.e. in the y-direction, increases with the increasing extent of the bed elevation discordance (Fig. 5c). This results in much smaller contribution of the I Tx component to the I T . The amount of reduc tion increases with the increasing dominance of the tributary flow, i.e. decrease in D R (Table 3).

The force of inertia I Tx is compared to its estimates based on the mean cross-sectional flow angle δ¯ (Hsu et al.) and the proposed σ correction coefficient value of 8/9 (Hager) in Figure 6. It is readily noticeable that the I Tx value is under predicted no matter which param eter is used for its estimation as long as Δz T ≤ 0.25h d . The use of the δ¯-angle results in either over prediction (Δz T = 0.50h d) or under prediction (Δz T ≤ 0.25h d) of I Tx . The over prediction does not exceed 10% and the under prediction does not exceed 25%. However, Table 3. The effect of D R and Δz T /h d on the percentage reduction of the I Tx contribution to I T when compared to the concordant beds' case. D R [/] Δz T /h d [/] 0.250 0.583 0.750 0.10 8.5 5.3 3.3 0.25 26.3 17.9 12.3 0.50 58.2 46.8 34.3 when the constant σ -value is used, the value of I Tx component is under predicted with more than 40%. The percentage reduction in some cases may reach 84%. Therefore, the use of the constant σ value is not recommended except in confluences with the greatest bed elevation discordance ratio when the tributary flow dominates ($\Delta z_T = 0.50h_d$ and $D_R = 0.250$). In such a case the percentage reduction is of the same order as that obtained with $\bar{\delta}$. Such a behaviour is a direct consequence of the already observed tendency of the σ-curve to approach proposed constant σ-value of Gurram et al. in the confluence with $\Delta z_T = 0.50h_d$ at low D_R values (Fig. 4b).

5 CONCLUSIONS

A 3D finite-volume based numerical model SSIIM2, which had been successfully applied in the previous study for estimation of 1D confluence model parameters in right-angled concordant beds' confluence, was used in this paper for estimation of these parameters in discordant beds' confluences with the same planform geometry. The comparison with the results for the concordant beds' confluence led to the following conclusions:

1. The increase in bed elevation discordance ratio $\Delta z_T / h_d$ reduces variation in the flow angle δ in the downstream tributary cross-section. Consequently, the mean cross-sectional flow angle $\bar{\delta}$ is increased.
2. The amount of this increase, when compared to the concordant beds' case, depends on the D_R-value. The greatest effect is achieved for $D_R = 0.750$, i.e. when the main-river flow dominates. The $\bar{\delta}$-angle in increased by 60% at maximum (i.e. from $\bar{\delta} = 33^\circ$ for $\Delta z_T = 0.00$ to $\bar{\delta} \approx 53^\circ$, for $\Delta z_T = 0.50h_d$). The maximum increase is successively halved for the remaining two D_R-values. For $D_R = 0.583$, the $\bar{\delta}$angle value of $\approx 50^\circ$, when $\Delta z_T = 0.00$, is increased to $\approx 65^\circ$, when $\Delta z_T = 0.50h_d$. This range is significantly narrowed when $D_R = 0.250$, i.e. $\bar{\delta} \in [66^\circ, 76^\circ]$ for $\Delta z_T \in [0.00, 0.50]h_d$.
3. Values of the correction coefficient σ are generally less than constant values of 8/9 and 0.85 proposed by Hager and Gurram et al., respectively. Moreover, they lay between the amended curve of Hager and that derived from experiments of Hsu et al. The only exception is the confluence with the greatest observed bed elevation discordance ratio ($\Delta z_T = 0.50h_d$) when the tributary flow dominates ($D_R = 0.250$). In this case the σ-value approaches the value 0.85 as proposed by Gurram et al.
4. The contribution of the tributary flow to the 1D momentum equation that is written for the main river direction reduces in the discordant beds'

confluences with the increase in bed elevation dis
cordance ratio, because of the reduced variation
of the flow angle $\bar{\delta}$ in the downstream tributary
cross-section.

5. The use of the mean cross-sectional flow angle $\bar{\delta}$
and the correction coefficient σ for the estimation
of the component of the tributary force of inertia
that acts in the direction of the main-canal flow may
lead to significant under prediction of the contri
bution of the tributary flow to the 1D momentum
equation. The under prediction of the I_{Tx} -value may
be as high as 84% when the constant σ -value of
Hager is used. Therefore, it is recommended to
use neither of proposed constant σ -values. Rather a
variable σ -value, deduced from the $\bar{\delta}$-angle value,
might be used, as the discrepancy between I_{Tx} and
$I_T \cos\bar{\delta}$ would not exceed 25%.

Best, J.L., and Reid, I. 1987. Closure to Separation zone at
open channel junctions, J. Hydraul. Eng., ASCE, 113, 4,
545–548.

Biron, P.M. & Lane, S.N. 2008. Modelling hydraulics and
sediment transport at river confluences. In: Rice, S.P., Roy,
A.G. & Rhoads, B.L. (Eds.) River confluences, tributaries
and the fluvial network. Wiley, 17–38.

Constantinescu, G., Miyawaki, S., Rhoads, B., Sukhodolov,

A. and Kirkil, G. 2011. Structure of turbulent flow

at a river confluence with momentum and velocity

ratios close to 1: Insight provided by an eddy-resolving

numerical simulation, Water Resour. Res. 47, 9W05507,

doi:10.1029/2010WR010018.

Đorđević, D. 2010. Numerical investigation of the river con

fluence hydrodynamics. Unpublished PhD Dissertation,

University of Belgrade, Belgrade, 382p Đorđević, D. 2013. Numerical study of 3D flow at rightangled confluences with and without upstream planform curvature. J. of Hydroinformatics, 15.4: 1073–1088. Đorđević, D. 2014. Can a 3D-numerical model be used as a substitute to a physical model in estimating parameters of 1D confluence models?, Proc. 3rd IAHR Europe Congress, Porto, 158–167. Gurram, S.K., Karki, K.S., and Hager, W.H. 1997. Subcritical junction flow, J. Hydraul. Eng., ASCE, 123, 5, 447–455. Gurram, S. K. and Karki, S. K. 2000. Discussion of Subcritical open-channel junction flow, J. Hydraul. Eng., ASCE, 126, 1, 87–89. Hager, W.H. 1987. Discussion of Separation zone at openchannel junctions, J. Hydraul. Eng., ASCE, 113, 4, 539- 543. Hager, W.H. 1989. Transitional flow in channel junctions, J. Hydraul. Eng., ASCE, 115, 2, 243–259. Hsu, C.C., Wu, F.S. and Lee, W.J. 1998a. Flow at 90 o equalwidth open-channel junction, J. Hydraul. Eng., ASCE, 124, 2, 186–191. Hsu, C.C., Lee, W.J. and Chang, C.H. 1998b. Subcritical open-channel junction flow, J. Hydraul. Eng., ASCE, 124, 8, 847–855. Lin, J.D. and Soong, H.K. 1979. Junction losses in open channel flows, Water Resour. Res., 15, 2, 414–419. Kennedy, B. 1984. On Playfair's law of accordant junctions. Earth Surface Processes and Lanforms, 9: 153–173. Olsen, N.R. 2000. CFD Algorithms for Hydraulic Engineering.Trondheim:The Norwegian University of Science and Technology. Olsen, N.R. 2012. A three-dimensional numerical model for simulation of sediment movements in water intakes with multi-block option – User's Manual, The Norwegian University of Science and Technology, Trondheim. Ramamurthy, A.S., Carballada, L.B. and Tran, D.M. 1988. Combining open channel flow at right angled junctions, J. Hydraul. Eng., ASCE, 114, 12, 1449–1460. Shumate, E.D., 1998. Experimental description of flow at an open-channel junction. Unpublished Master thesis, Univ. of Iowa, Iowa, 150 p. Taylor, E.H. 1944. Flow characteristics at rectangular openchannel junctions, Transactions, ASCE, 109, 893–912. Weber, N.B. and Greated, C.A. 1965. An investigation of flow behaviour at the junction of rectangular channels, Proc. Instn. of Civ. Eng., Thomas Telford Ltd, London, 34, 321–334. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Practical application of numerical modelling to overbank flows in a compound river channel

P.M. Moreta & A.J. Rotimi

School of Computing and Engineering, University of West London, Ealing, London, UK

A.S. Kawuwa

Faculty of Engineering and Physical Sciences, University of Surrey, Guildford, UK

S. Lõpez-Querol

Department of Civil, Environmental and Geomatic Engineering, University College of London, Ealing,

London, UK

ABSTRACT: Compound sections formed by a river channel and floodplains, are used in river channels design to provide additional conveyance capacity during high discharge periods. When the overbank flow occurs, the flow in the river channel is affected by the momentum transfer between the main channel and floodplains, which modifies water levels and velocity distributions given by traditional methods. One-dimensional (1D) models using the Single Channel Method (SCM) and the Divided Channel method (DCM) have been proven to be not accurate enough in compound channel flows. New more advanced models have been developed in order to accurately estimate discharge flows and depth-averaged velocity distributions. The quasi-two dimensional

model Conveyance Estimation System (CES) estimates discharges and velocities in a cross-section based on the

Lateral Distribution Method (LDM). Two-dimensional (2D) modelling solves the depth-averaged Navier-Stokes

equations in a discretized reach of a river. In this work, published field measurements of the River Main (UK) are

analyzed via 1D, CES and 2D modelling, in order to find a practical solution to give good predictions of water

levels and velocity distributions in overbank flows in river channels with floodplains. The results show that 1D

modelling combined with CES gives reasonable accurate values and is a complementary tool for advanced 2D

models in real conditions.

1 INTRODUCTION

World population growth has gradually resulted in increased human settlements, developments and activities around the floodplains of rivers which lead to disastrous effects during flooding of natural rivers. River floods result in huge losses in human lives and economic losses. A third of the world's losses due to natural disasters is caused by flood disasters, flooding also accounts for half the loss of life with analyses of the trend showing that this figures have significantly increased (Berz, 2000). Accurate estimation of flow rate in channels is of enormous significance for flood prevention. Flooding occurs when the quantity of water flowing along a channel is higher than its carrying capacity. Hence the need for accurate pre diction of river discharges during flood conditions to mitigate the impact, thereby saving lives and prop erties has drawn greater attention of researchers and engineers in recent times. There are numerous meth ods and approaches that have been employed in recent times to facilitate accurate estimation and prediction of discharge, conveyance and water surface level of rivers during overbank flow. Previous work in compound open channels has been mainly focused on modelling uniform flow conditions and compared with experimental data in laboratory flumes. Shiono and Knight (1991) developed a quasitwo dimensional model (based on lateral distribution method) to model conveyance in compound cross sections. This approach has been used in the Environment Agency's Conveyance Estimation System (CES). Mc Gahey et al (2008) demonstrated the ability of CES to accurately estimate lateral velocity distribution and discharges assuming uniform conditions in real rivers. The aim of this work is to validate the application of one-dimensional Lateral Distribution Method (LDM) via the use of Conveyance Estimation System (CES) which is a commercial software for the estimation of discharge/conveyance capacity of compound channels and compare the results to that of the traditional onedimensional methods, Single Channel Method (SCM) and Divided Channel Method (DCM) using Hydrological Engineering Centre River Analysis System (HEC-RAS) software. The two-dimensional SRH-2D (Sedimentation and River Hydraulics 2 Dimensional) model is also used for comparison of velocity distributions.

The data that were utilized for successfully car rying out the simulation in this research work were obtained from the previous work of Martin and Myers (1991), Myers and Lyness (1994); Lyness and Myers (1994a) and Lyness and Myers (1994b), conducted on river Main, Northern Ireland. The aim of the project was achieved by simulation of the study reach of river Main (which is a reconstructed prototype river reach in Northern Ireland, United Kingdom) on both CES and HEC-RAS computational modelling software and the two-dimensional code SRH2D. The three codes have been applied by using the same boundary conditions, cross-section data and flow parameters in order to have the same criteria for comparison and validation. Finally the water surface level and velocity distribu tion results obtained from this software were analysed and compared with available field data to validate and verify the results.

A great effort has been made over the last decades to improve calculation of water levels and velocities in real rivers by the use of 2D and 3D modelling. However some important uncertainties are still unsolved. In this context, an accurate 1D model easy to calibrate and with the support of the CES can be an improved tool for comparison.

2 LITERATURE REVIEW

In compound channels, the velocity gradient between the main channel and floodplain flows generates shear forces in the main channel-floodplain interfaces. Sellin (1964) presented photographic evidence of the bank horizontal vortices acting along the interface and together with Zheleznyakov (1971) demonstrated a decrease in the main channel discharge after over bank flow occurs, only partially compensated by some discharge increase on the floodplain. The physics of flood hydraulics has been widely studied during the last 30 years (Knight and Shiono, 1996; Sellin, 1996 and Wormleaton et al 2004), concluding in a deep knowledge and understanding of the phenomenon involved.

Commercial models, such as HEC-RAS and MIKE 11, use calculation methods like SCM and DCM. The SCM considers the same velocity for the whole sec tion.The DCM separates the cross-section into areas of different flow characteristics, such as the main channel and floodplains. Wormleaton et al. (1982) demon strated that the SCM underestimates the conveyance capacity and the DCM overestimates compound chan nel. In the following years, several researchers pre sented some improved methods for compound channel flow estimation, Wormleaton and Merret (1990) pro posed a simple modification that improves the DCM estimation and the DCM was empirically corrected by Ackers (1992). An alternative and more advanced method was developed in those years, the lateral dis tribution method (LDM) formulated by Wark et al.

(1990) and the method by Shiono and Knight (1991).

These two methods are based on the same equations and calculate the lateral velocity distribution in the cross section, like a quasi-2D model. This paper aims to discuss refinements in 1D modelling that are able to cope with such complexities in a straightforward way. The research focuses on the prediction of the velocity distribution across the river. While the free surface profile is computed reasonably well by 1D numerical models, the same does not hold for the velocities unless the appropriate term at the interface is used. The methodology presented herein uses the HEC-RAS and the CES in order to improve 1D numerical modelling. The interaction between the main channel and the floodplain is modelled by using the lateral distribution of velocities given by CES. This method is applied to previously published field data from River Main. Moreover, the results given by widely used 2D models (SRH2D), will be used for comparison. 3 RIVER MAIN FIELD DATASET The river data under study consist of a reach of the river Main, in Northern Ireland, which has some length of its reach reconstructed and realigned (between 1982 and 1986). This reach of the river comprises a trapezoidal compound channel with a centralised deep main channel bordered by one or two side berms. Numerous number of research works have been carried out on this river reach (Martin and Myers, 1991; Myers and Lyness, 1994; Lyness and Myers, 1994; Defra/Environmental Agency, 2003), with the aim of having a better understanding of the hydraulic behaviour of two-stage waterways. The measured study reach is found to have a longitudinal length of 800 meters from upstream (section 14) to downstream (section 6) with an average longitudinal bed of 0.003 or 1:520 with flood plains slope towards the main channel having a gradient of 1:25. It is divided into nine cross sections, situated at equal intervals of 100 meters apart. The plan view, upstream and the downstream cross sections of the river Main reach under investigation are shown in Figures 1 and 2. The river bed material comprises of a very coarse gravel with a D 50 size ranging between 100 and 200 mm. Quarried stones of up to 0.5 tonne in weight and having a size up to 1 m in diameter are used as a rip-rap to protect the side slopes of the main channel. The grass and weed that cover the berms are maintained regularly by keeping them short (Martin and Myers 1991). Figure 3 shows a cross-sectional view of the compound river channel and the bed materials in floodplains and river banks. Some typical water surface profiles measurements obtained using steady flow computation for the discharges of 10.5, 20.1 and 51.3 m 3 /s were shown by Myers and Lyness 1994 and reproduced in figures 5 and 6 in the next section. The 10.5 m 3 /s discharge corresponds to an inbank flow and the two higher discharges (20.1 and 51.3 m 3 /s) are overbank flows, the lower under the top floodplain level and the higher full

Figure 1. River Main plan view. Location of cross-sections

from upstream (s14) to downstream (s6).

Figure 2. Upstream (doted) and downstream (full line) cross

sections, numbers 14 and 6 respectively, of River Main reach

of study.

Table 1. Main geometric and hydraulic parameters in the

River Main study reach. Upstream s14 Downstream s06

Long.. Bed slope 0.0052 0.0019

Bankfull flow 20.1 11-12

Manning n (m.c.) 0.39 0.39

Manning n (f.p.) 0.40 0.40

Bed width 12.2 11.1

Lateral slope (f.p.) 1:25 1:20

covering the floodplains. Martin and Myers (1991),

Lyness and Myers (1994b) and Lyness et al (1987)

indicate that SCM underestimate discharges, while

that of DCM overestimates it, revealing there is an

exchange of momentum between floodplains and main

channel, in overbank flow situation. Figure 3. River Main: river channel and floodplains. Main channel is covered by cobbles, with medium rip-rap stones for the bank slopes and floodplains with natural grass. Table 2. Geometry in River Main sections. Upstream S14 Downstream S06 Y (m) Z (m) Y (m) Z (m) 0.0 40.40 0.0 38.00 5.3 37.81 7.1 35.32 13.5 37.32 13.6 34.82 14.4 36.40 14.9 34.00 26.6 36.40 26.0

34.00 27.6 37.38 27.3 34.87 35.7 37.78 34.4 35.37 40.8 40.40 39.5 38.00 Table 3. Slope and distance between cross-sections. Section Distance Bed Level Bed slope 14-13 100 36.40 0.0055 13-12 100 35.85 0.0032 12-11 100 35.53 0.0031 11-10 100 35.22 0.0012 10-9 100 35.10 0.0058 9-8 100 34.52 0.0017 8-7 100 34.35 0.0018 7-6 100 34.17 0.0017 6-… 0 34.00 0.0030 Roughness Manning's n coefficients were estimated by using uniform flow conditions in upstream crosssection 14 by Martin and Myers (1991) and Myers and Lyness (1994a). These studies found that using Manning's formula the inbank roughness decreased between n = 0.050 for low depths and n = 0.039 for bankfull. However the Manning's n for the gravels/cobbles in the bed varies between n = 0.025–0.039, and the Manning's n for the riprap on the bank is higher than n = 0.040. This means that as the water depth increases the main channel mean roughness should be greater, which does not fit the estimation of Manning's roughness by using the mean bed slope and field rating curves.

Figure 4. River Main bed profile and water level profiles measured in fieldworks for inbank (10.5), and overbank (20.1 and 50.3) flows (after Myers and Lyness 1994).

4 NUMERICAL MODELS

In the next subsections the models used in the present work are briefly described. The 1D HEC-RAS model (USACE, 2008), the CES model (Environment Agency, 2004), and the SRH2D (Lai, 2008).

4.1 HEC-RAS 1D model

The results obtained with 1D modelling based on the energy or Bernoulli equation (HEC-RAS) are com pared here with the field measurements in terms of free surface profile and velocity distributions. The DCM and SCM were used by applying the HEC RAS model, as well as CES in backwater computation mode. Under steady conditions, the one dimensional hydraulic equations to be solved are the conservation of mass:

and the conservation of energy:

where A = cross-sectional area normal to the flow; Q = discharge; g = acceleration due to gravity; H = elevation of the water surface above a specified datum, also called stage; S_o = bed slope; S_f = energy slope; x = longitudinal coordinate. Equations (1) and (2) are solved using the well known four-point im plicit box finite difference scheme (USACE, 2008). HEC-RAS solves these equations using the standard

step method as follows: where Y_i = depth of water at cross-sections; Z_i = elevation of the bed; V_i = average velocities at cross-sections; α_i = velocity weighting coefficients; h_e = energy head loss. The energy head loss can be calculated multiplying the length between the crosssections times the friction slope, S_f . HEC-RAS uses two methods for computing the value of S_f , depending on whether the cross-section is treated as a unique compound section (SCM) or it is divided in sub-sections (DCM). The equations for the SCM are: where R = hydraulic radius of the whole section, K = hydraulic conveyance, and n = Manning's roughness coefficient for the whole section. The DCM divides the cross-section into a main channel and two lateral floodplains applying eq. (3) for each subdivision and calculating the friction slope separately: where the subscript i differentiate the three subsections. The total $K = \sum K_i$ and the total $Q = \sum Q_i$. HEC-RAS software implements the Flow Distribution Option in order to compute the lateral velocity distribution V_{di} by dividing the cross-section into a number of slices and then calculating the V_{di} as: where A_{si} = cross-sectional area for each slice; Q_{si} = discharge for each of the slice. 4.2 CES quasi-2D model The Environment Agency's CES model is based on the LDM (Wark et al, 1988; Shiono and Knight, 1991, Ervine et al, 2000), and it combines the continuity and momentum depth-averaged equations of motion for steady conditions and in the stream-wise component. The general equation of the model for a straight river (sinuosity equal 1.0) is obtained: where f = Darcy's friction factor; q = streamwise unit flow rate (=Y·U d); U d = depth-averaged velocity; S y = lateral bed slope; λ= non-dimensional Boussinesq eddy viscosity; y = lateral horizontal coordinate;

and $ = secondary flow parameter. The first term in eq. (3) is the hydrostatic pressure, the second is the boundary friction term, the third is the turbulence due to lateral shear stress and the last term in the right side represents the secondary circulations. The recom mended values for the different variables and the Finite Element Code for solution of Equation 3 can be found in Defra/EA 2003. Once the velocity in each slice, U d , is obtained, the total discharge, Q t , in the cross section can be calculated as sum of unit discharges as:

where y i and y i−1 are the horizontal coordinates in transverse direction for both sides of the slice.

4.3 SRH2D model

The two dimensional depth-averaged model SRH 2D is a free-use available numerical code developed by Yong G. Lai, from U.S. Bureau of Reclamation (Lai, 2010). The code is based on the finite-volume approach and it can be assumed that provides an acceptable solution of the 2D equations in a variety of river flows (Lai, 2000; and Lai et al, 2006). The model solves the shallow water equations of flow:

Continuity equation:

Momentum equations in x and y:

where, x and y are horizontal Cartesian coordinates;

z_b is bed elevation, t is time; H is water depth; U d

and V b are depth-averaged velocity components in x

and y directions, respectively, τ_{xx} , τ_{xy} and τ_{yy} are

depth-averaged stresses due to turbulence as well as

dispersion, ρ is the water density, and τ_{xb} , τ_{yb} , are

the bed shear stresses. These bed stresses are obtained

using the Manning's resistance equation as follows: Figure 5. River Main, mesh discretization (15680 elements). The mesh is denser in the main channel banks, inclined bed, and less dense where the bed is flat. where C_f is a friction coefficient that is mainly depending on n, the Manning's roughness coefficient. The turbulence stresses are computed with Boussinesq equation as: where υ is kinematic viscosity of water and υ_t is eddy viscosity. The eddy viscosity is calculated with the k-ε turbulence model (Rodi 1993), and the eddy viscosity is calculated as: with two additional equations for the turbulent kinetic energy, k , and its dissipation rate, ε. Launder and Spalding (1974) added two transport equations to solve the new two unknown variables. The numerical solution of the SRH2D equations is implemented in a finite-volume method with quadrilateral elements. The standard conjugate gradient solver with ILU preconditioning is used (Lai 2000) for spatial integration in an iterative process. The computational domain is discretized in 15939 nodes and 15680 quadrilateral elements, 99 in cross-stream direction and 160 in streamwise, as it is shown in Fig. 5. The boundary conditions are total discharge at the upstream section and a unique mean water level for the

downstream cross-section. SRH2D calculates a distri

bution of the velocity along the upstream condition in

such a way that the total discharge is satisfied. The

approach used in this work is the conveyance dis tribution, which at the inlet is distributed across the upstream section following the conveyance proportion, eq. (7), so then the velocity in each element is proportional to depth and inversely proportional to Manning's n. This approach overestimates velocities in the main channel and underestimates them in the floodplain. These boundary conditions are the same than the 1D model, so the differences in results are only dependent on modelling equations.

5 MODELLING APPROACH

The first step in a river modelling work once the topography and geometry is defined is to identify the hydraulic variables involved. The most important one is the hydraulic resistance to flow, defined in terms of Manning's roughness coefficient in this work. In order to reduce the uncertainty explained in previous sections about the Manning's n value, this is calibrated with the 800 m longitudinal water level profile in the 10.5 m^3/s inbank discharge. The HEC-RAS 1D model is used for iterating different Manning's n and to obtain the bankfull n by fitting the computed water profiles with the field profiles. Figure 6 shows the computed profile obtained with a mean Manning's n = 0.041 in the main channel, which is the roughness that best fits the field data. According to the variation of roughness with depth, the Manning's n in the bed should be 0.045 and 0.030 in the banks.These values will be used as the main channel roughness coefficients in the overbank discharges.

The water levels obtained with HEC-RAS for the two overbank discharges (20.1 and 51.3 m^3/s) are shown in Fig. 7. The bank stations are located on the top of the main channel (DCM) or on the top of the floodplain walls (SCM) and two separated solutions are obtained.The results illustrate the main differences between both methods. The DCM gives lower water levels than the SCM for the same discharge. In gen eral the water levels measured in the field works are found between the two numerical solutions (DCM and SCM). Some small discrepancies appear, probably due to roughness variation with depth and/or local changes in slope or section area.

Previous studies in compound channel flows demonstrated that DCM and SCM are not providing good results under uniform flow conditions. In this paper, Fig. 7 demonstrates that for gradually variable flow conditions DCM and SCM maintain discrepan cies in water levels with real data and give different results. In terms of velocity distribution across the section, the SCM gives a uniform velocity for the whole section and the DCM is not providing a real distribution. For overbank flow the velocities given by HEC-RAS are different to the real distribution, especially in the main channel. Fig. 8 shows the Figure 6. Field (9 full dots) and computed water surface profiles by using HEC-RAS with the estimated (DCM-nEST) and calibrated Manning coefficients (Manual) for the inbank flow. Figure 7. Computed water surface profiles by using HEC-RAS (DCM and SCM) for the two overbank discharges. Comparison with field data (Myers and Lyness, 1994). Figure 8. Field measured and 1D computed velocities in section s14 for inbank and overbank discharges, Q10.5 and Q51.3. velocity distribution obtained with HEC-RAS for the inbank flow Q10.5 and for the overbank flow Q51.3, together with the values measured for Q51.3. The model overestimates velocities in main river banks. In order to understand the flow behaviour in this gradually varied flow river, the SRH2D was applied to the computational domain in Figure 5. The way the 2D model estimates total energy is a combination of bed friction (through a roughness coefficient)

Figure 9. Water level field data and 2D computed water surface (W .S.) compared with 1D values (DCM). Inbank discharge Q10.5.

Figure 10. Water level field data and 2D computed water surface (W .S.) compared with 1D values (DCM). Overbank discharges Q20.1 and Q51.3.

and turbulence stresses (through a dissipation coeffi cient). This is an important advantage with respect to 1D modelling that only includes bed friction losses and the velocity distribution only depends on water depth and Manning's coefficient. The results obtained with SRH2D model are shown in Figs 9 and 10. The water level profiles obtained with 2D modelling (k-ε turbulence model) are lower than those obtained with the DCM for all the discharges. The Manning's coef ficients are the same in both models, as well as the boundary conditions. The difference in water levels between 2D and 1D solution are smaller for overbank flows than for the inbank one. This result confirms the conclusions by Moreta (2014) who demonstrated that for uniform flow in straight compound channels, 2D modelling gives lower water levels than 1D modelling if the roughness and boundary conditions are the same. However 2D modelling has some advantages over 1D modelling. First, the changes in main channel and floodplain sinuosity are taken into account, second, it considers internal energy losses due to flow turbu

lence and third, consequently the velocity direction Figure 11. Velocities of field data (Martin and Myers, 1991) and computed with 1D (DCM), 2D (SRH) and CES for overbank discharge Q51.3. Figure 12. Velocities of field data (Martin and Myers, 1991) and computed with 1D (DCM), 2D (SRH) and CES for overbank discharge Q51.3. and distribution must be better simulated. In Figures 11 and 12 the velocity distribution obtained with 1D (DCM) and 2D models for the inbank, Q10.5, and overbank, Q51.3, discharges are compared with field measurements. The velocities given by SRH2D improve slightly the velocities obtained by DCM. 6 IMPROVING 1D MODELLING In order to improve 1D modelling (with DCM), the results obtained with CES are discussed in this paragraph. The first step is that for straight river channels with moderate roughened floodplains, the water profiles obtained by 1D model are better than the 2D model. However, the distribution of depth-averaged velocity can be obviously improved. The CES is applied to section 14, using the same bed slope and Manning's coefficient of roughness than in 1D modelling. CES precise a water level to estimate the velocity distribution and total discharge. The water depth used for estimating the velocity is that obtained from the 1D modelling. Figures 11 and 12 show that the velocity distribution obtained with CES fit better with the data than the distribution given by 2D model. 7 CONCLUSIONS The numerical analysis of this work is based on previously published field data and illustrates some of the problems that affect common 1D numerical model in reproducing overbank flow. HEC-RAS model is not able to yield an accurate velocity distribution across the section of a straight compound channel. Secondly, the comparison between the field data and the SRH2D model shows the need to take into account that the Manning's coefficients valid for 1D modelling are not enough accurate for 2D simulations. Therefore, some uncertainties rising from the use of 2D models can provide uncertain results respect to better predictable estimations obtained by 1D modelling.

The analysis and comparison of flow velocities measured in field works and computed by numerical models has shown that the prediction of accurate veloc ity distributions in compound channel flow is a major challenge in numerical modelling. Typical 2D finite volume codes based on k-ε turbulence model trend to under predict main channel and floodplain interaction. These 2D models slightly improve the depth-averaged velocities obtained with 1D model for the straight river case analysed herein. In order to better simulate veloc ities, the CES based on Lateral Distribution Method is proposed for comparison. The CES gives a better rep resentation of momentum interaction between main channel and floodplains and of the velocity distribu tion across the section. This methodology has been contrasted with field river data under gradually varied conditions, confirming the results of some previously published works on the topic under different condi tions (Weber and Menendez, 2004, and Vionnet et al, 2004).

Ackers, P. (1992). “Hydraulic design of two-stage channels.” Proc. ICE, Water Maritime and Energy, London 96(4), pp. 247–257.

Babaeyan-Koopaei, K. Ervine, D.A., Sellin, R.H.J. (2001). “Development of a UK data base for predicting flood levelsfor overbank flows”. Environment Agency of UK.

Berz, G. (2000). ‘Flood Disasters: Lessons from the past worries for the future’. Proc. Inst. Of Civil Engineers: Water, Maritime and Energy, London, March, 142 (1).

Knight D.W. and Shiono K. (1996). “River channel and floodplain hydraulics”. In Floodplain Processes, (Eds. Anderson, Walling & Bates), Chapter 5: 139–181. J Wiley.

Defra/Environment Agency (2003) “Reducing uncertainty in river flood conveyance, Interim report 2 – Review of meth ods for Estimating Conveyance”. Project W5A-057, HR

Wallingford, UK.

Lai,Y.G. (November, 2008)Theory and user manual for SRH

2D. Denver, Colorado: U.S. Bureau of Reclamation.

Lai, Y.G. (2009) Two-Dimensional Depth-Averaged Flow

Modeling with an Unstructured Hybrid Mesh. Denver,

Colorado: U.S. Bureau of Reclamation. Lai, Y.G. (2008) Sedimentation and River Hydraulics software (SRH-2D). Version-2 ed. Denver, Colorado: U.S. Bureau of Reclamation. Lyness, F.J. & Myers, W.R.C. (1994a). “Velocity coefficients for overbank flows in a compact compound channel and their effects on the use of one dimensional flow model”. 2nd Int. Conf. on River Flood Hydraulics,York, UK, 22–25 March, John Wiley & Sons Ltd. p.379–398. Lyness, J.F. and Myers, R.C. (1994b). “Comparisons between measured and numerically modelled unsteady flows in a compound channel using different representation of friction slope”. 2nd Int. Conf. on River Flood Hydraulics, York, UK, 22–25 March, John Wiley & Sons Ltd. Martin L. A. and Myers, W.R.C. (1991) “Measurement of overbank flow in a compound river channel”. Proc. ICE, Water Maritime and Energy, 2, p.645–657. Martín-Moreta, P. (2010). “2-D model for vegetated rivers; calibration with Besòs river data.” (in Spanish) PhD thesis, UPC, Barcelona, Spain. McGahey, C., Samuels, P.G. and Knight D.W. (2006). “A practical approach to estimating the flow capacity of rivers – application and analysis”. Proc. River Flow 2006, Lisbon, September, 1: 303–312. Myers, W.R. and Lyness, F.J. (1994). “Hydraulic study of a two-stage river channel”. Regulated Rivers Research and Management, 9(4), p.225–235. Sellin, R.H.J. (1964). “A laboratory investigation into the interaction between the flow in the channel of a river and that over its flood plain”. La Houille Blanche. 7, pp. 793–802. Shiono, K., Knight, D.W. (1991). “Turbulent open channel flows with variable depth across the channel.” J. Fluid Mechanics, 222, pp. 617–646. US Army Corps of Engineers, “HEC-RAS, Hydraulic Reference Manual,” Hydrologic Engineering Center, Davis Version 4.0, 2008. Vionnet, C., Tassi, P., Martín-Vide, J.P. (2004). “Estimates of flow resistance and eddy-viscosity coefficients for 2D modelling on vegetated floodplains.” Hydrological processes, Vol. 18, Issue 15, pp. 2907–2926. Wark, J.B., Samuels, P.C., Ervine, D.A. (1990). “A practical method of estimating velocity and discharge in compound channels.” Proc. River Flood Hydraulics pp. 163–172. Weber, J.F. & Menéndez, A.N. (2004) Performance of lateral velocity distribution models for compound channel sections. International Conference on Fluvial Hydraulics, 2. Wormleaton, P.R. Allen J., Hadjipanos P. (1982). "Discharge assessment in compound channel flow". Journal of Hydraulics Division, ASCE, 108(9), pp. 975–994. Wormleaton, P.R., Merrett, D.J. (1990). "An improved method of the calculation for steady uniform flow in prismatic main channel/flood plain sections". Journal of Hydraulic Research, Vol. 28, n o 2, pp. 157–174. Wormleaton, P.R., Sellin, R.H.J., Bryant, T., Loveless, J.H., Hey, R.D., Catmur, S.E. (2004). "Flow structures in a twostage channel with a mobile bed". Journal of Hydraulic Research, Vol. 42(2), pp. 145–162. Zheleznyakov G.V. (1971). "Interaction of Channel and Floodplain Streams". Proc. 14th IAHR congress, Paris, vol. 5, pp. 145–148. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Comparison between different methods to compute the numerical fluctuations in path-conservative schemes for SWE-Exner model

Francesco Carraro, Valerio Caleffi & Alessandro Valiani

Dipartimento di Ingegneria, Universitá degli Studi di Ferrara, Ferrara, Italy

ABSTRACT: We consider the Shallow Water Equations (SWE) coupled with the Exner equation. To solve these balance laws, we implement a P_0P_2-ADER scheme using a path conservative method for handling the non-conservative terms of the system. In this framework we present a comparison between three different Dumbser-Osher-Toro (DOT) Riemann solvers. In particular, we focus on three different approaches to obtain the eigensystem of the Jacobian matrix needed to compute the fluctuations at the cell edges. For a general formulation of the bedload transport flux, we compute eigenvalues and eigenvectors numerically, analytically and using an approximate original solution for lowland rivers (i.e. with Froude number $Fr \ll 1$) based on a

perturbative analysis. To test these different approaches we use a suitable set of test cases. Three of them are presented here: a test with a smooth analytical solution, a Riemann problem with analytical solution and a test in which the Froude number approaches unity. Finally, a computational costs analysis shows that, even if the approximate DOT is the most computationally efficient, the analytical DOT is more robust with about 10% of additional cost. The numerical DOT is shown to be the heavier solution.

1 INTRODUCTION

Nowadays one of the most challenging issue in geo physics and civil engineering is to understand how the environmental changing influences human activities.

In this field, morphodynamical models are extensively used to predict the evolution of river and coastal environments quantifying the interactions between sediment transport and water flow.

A large part of this kind of mathematical mod els, especially for the simulation of natural rivers and nearshore hydrodynamics, is based on the Shallow Water Equation (SWE) coupled with the sediment bal ance equation due to Exner (11). These models have been studied with two different approaches: splitting the computation of the hydrodynamics from the bot tom topography adaptation or solving the full coupled system of equations.

The splitting method consists in solving separately the SWE for a fixed topography, and updating after wards the topography using the Exner equation (e.g., (17)). Although splitting methods had been widely used, Cordier et al. (7) showed why this strategy could generate some unphysical oscillations related to the hyperbolicity of the system of equations.

On the other hand, the fully coupled approach present a critical issue too. Because of the full cou pling of the equations, the variable bottom topog raphy reveals a non-conservative product ((20) and its developments, e.g. (18)). In order to handle such non-conservative term, great interest has been devoted

on the path conservative (or path consistent) method developed by Parés (18). In particular, an Osher Riemann Solver was proposed by Dumbser and Toro (10). The non-conservative formulation of the SWEExner system may compromise the uniqueness of the solution of the Rankine Hugoniot condition through shock waves (1), even though Castro et al. (5) concludes that the path conservative method coupled with physical considerations (e.g., the Lax-Friederichs entropy condition) ensures a formally consistent numerical solution. Moreover, the uncertainty of the solution may be noticeable for very fine meshes, discontinuities of large amplitude, or large-time simulations (13). An example of how the path consistent method coupled with physical considerations provides good results has been recently presented in (4). Here we study the application of the path-conservative method to the morphodynamic problem, with particular attention on the numerical efficiency of the model. To do that we implement a one step P N P M ADER scheme. This numerical model, proposed by Dumbser et al. (8), is an extension of the one step ADER approach due to Toro et al. (19). It unifies the construction of one-step finite volume and discontinuous Galerkin schemes and ensures great versatility and numerical efficiency. This paper is organised as follows: in section 2 the structure of the ADER model is summarized, focusing on three different implementations of the Riemann solver; in section 3 numerical results are shown; in section 4 a computational costs analysis is presented; finally, in section 5, conclusions are drawn.

2 NUMERICAL MODEL

In the present work, the system of equations that is solved consists of the classical one dimensional SWE coupled with the Exner equation, for the hydrodynamics and for the bed evolution, respectively.

The suspended sediment load transport is assumed to be negligible with respect to the bed load sediment transport. In the source term, only the effect related to the bottom topography is considered. In this work, in order to perform a more tight validation of the model, the friction between the flow and the bottom is not considered, even though the computation of the solid discharge is carried out using the shear velocity. Thus, the resulting system of equation is a quasi-linear hyperbolic system:

in which W is the vector of the conservative variables and A(W) is the Jacobian of the flux, including the non conservative product, i.e.:

being $y(x, t)$ the water depth; $q(x, t)$ the specific water discharge; $z(x, t)$ the bed elevation; $u = q/y$ the depth averaged velocity; $c = \sqrt{g y}$ the propagation celerity of gravitational waves (g is the gravity acceleration); $\xi = 1/(1 - p)$ and p the porosity of the riverbed; q_s the solid transport discharge (in volume).

In order to solve (1), we implement a 1D, P_0P_2

ADER scheme using the path conservative method for

handling the non-conservative term due to the movable

bottom topography.

update of W from time t^n to time $t^n + \Delta t$, the whole

algorithm is structured in four parts.

Reconstruction. Apply the WENO scheme of arbi

trary accuracy with non-negative weights proposed in

(9) to the numerical solution W at time t^n represented

by piecewise constant state to obtain the parabolas

$\hat{w}^n_p = \hat{w}(W(t^n))$.

Local data evolution. Use the reconstructed solution

$\hat{w}^n_p$ in each element as initial condition for the predictor

iterative scheme, i.e. the local continuous space-time

Galerkin method presented in (8).

Compute the fluxes at cell interfaces. Solve the Rie

mann problem at the cell interfaces approximately

using the space-time polynomials of the state gen

erated in the previous step. In this work we used

the Dumbser-Osher-Toro (DOT) Riemann solver pro

posed by Dumbser and Toro (10) to approximate the

flux with the Osher jump $D^{\pm}_{j\pm\frac{1}{2}}$.

According to them, we remark that the DOT solver is an Osher-type Riemann solver, i.e. it uses the full eigensystem of the hyperbolic system and thus, an individual – and in general different – numerical viscosity is attributed to each characteristic field. This type of solution is named complete. In order to properly treat the non conservative term, we use the path conservative scheme to compute the Roe-matrix A that satisfy the generalized RankineHugoniot condition: where ! (W – j+ 1 2 , W + j– 1 2 , s) is a general path, W ± j+ 1 2 are the point values at the cells edge and s is the curvilinear coordinate along the path. The main task of this paper is to compare three different methods to compute the integral on the right hand side of equation (3), in the particular case of simple segment path. Update. Update the solution from time level n to time level n + 1 in a one step, e.g. [8, pp. 8220-8221]. 2.1 Solution of Riemann problems We focus our attention on the computation of the integral on the right hand side of equation (3), i.e. on the matrix | A|. Adopting a G-point quadrature rule in the unit interval [0; 1] and introducing a simple segment path !(s) = W – + s(W + – W –), equation (3) becomes: in which w Gi are the weights of the quadrature rule and for the absolute value operator of a matrix the usual convention applies: where R is the matrix of right eigenvectors of A, R –1 is its inverse and |�| is the diagonal matrix of the corresponding eigenvalues of A in absolute value. Equation (4) shows that the DOT scheme requires the eigensystem of the matrix A several times during the numerical integration along the path: in general, if the eigensystem is computed numerically, it can be a relevant source of computational cost. However, in the particular case of the SWE-Exner model, the numerical one is not the only way to get the eigensystem of matrix A in equation (2): in literature there is an analytical solution for this problem due to Castro et al. (6);

but, also, we present here a new numerical approxima

tion of eigenvalues and eigenvectors for SWE-Exner

models. Thus we define: numerical DOT the Rie

mann solver that computes |A| using the numerical

calculation of the eigensystem of the Jacobian matrix;

analytical DOT the one that adopts the formulation

proposed in (6) and resumed in section 2.2; approxi

mated DOT the Riemann solver in which we introduce the approximate estimation of the eigensystem of A

that we present in section 2.3.

2.2 Analytical formulation of eigenvalues and eigenvector for SWE-Exner models

In this section the analytical eigensystem of A intro

duced by Castro et al. (6) is summarized. The formu

lation is written in a non-dimensional form, to point

out the physical meaning of the eigensystem.

In order to obtain a non-dimensional formulation

of the eigenvalues of matrix A(W), the characteristic

polynomial can be divided by c^3 . In this way, the solu

tion proposed in (6) (also correcting a typing error)

becomes:

where Fr= u/c is the Froude number of the water

flow, $k_2 = Fr^2 + 3 + 3\delta^*_q$, $\varphi = \arccos(k_1 / \sqrt{4k_2^3})$, $k_1 =$

$27\delta^*_y + 18Fr(1 + \delta^*_q) - 2Fr^3$, being δ^*_y and δ^*_q the non

dimensional quantities $\delta^*_y = \xi\, c\, \partial q_s\, \partial y$ and $\delta^*_q = \xi\, \partial q_s\, \partial q$.

The associate right eigenvectors are

This formulation emphasizes that the properties of

the system (1) depends only on i) the Froude number

of the flow and ii) the relation adopted for the solid

transport discharge. The study of the hyperbolicity of

the system will be treated in section 2.4.

2.3 A new approximation of eigenvalues and eigenvector for

SWE-Exner models

In most lowland rivers, because of the subcritical

regime of the water flow (i.e., Fr� 1), the sediment

transport is very small. Thus, let us define a general

form of the sediment transport relation q s dependent

from a “small parameter” K : where θ is the non dimensional shear stress on the riverbed grains; θ c is the threshold value of the shear stress θ ; U 0 and Y 0 are a reference velocity and a reference depth of the flow; a s and b s are two parameters of the bed load sediment transport. Adopting the Meyer Peter & Müller formula a s can be set equal to 8 (15) or 4 taking into account the correction proposed by Wong & Parker (21); while b s is usually set equal to 3/2. Looking at the non dimensional shear stress, θ c represents an empirical constant value, usually set equal to 0.045; while θ (using a power-law resistance formula) can be written as: being u ∗ the shear velocity: where C 0 = 7.66 is an empirical constant; S g is the rate between the sediment density ρ s and the water density ρ; d m is the mean diameter of the sediment grain in the bed, while e s = 2.5 · d m is an approximation of the equivalent roughness height. In this framework, the small parameter K can be defined as: For a typical sandy riverbed – e.g., with p = 0.4, S g = 2.6, d m = 5 · 10 −4 [m] and U 0 Y 0 = 1 [m 3 /(s m)] – from equation (11) results that K = 7.4 · 10 −5 1. Adopting the general formulation (8) for the load sediment transport, the Jacobian matrix of system (1) becomes: The structure of eigenvalues and eigenvectors of the matrix A^ can be investigated considering that the non dimensional parameter K is small. Thus, for the eigenvalues the Perturbative theory applies: where λ (0) i indicates the i − th exact eigenvalue of the classical clear water problem:

while λ (1) i represents the first order terms of the i − th

eigenvalue of the full matrix A^:

Moreover, the right eigenvectors matrix is:

with:

A s = u 2 − c 2 c 2 − K 2U u 2 − c 2 ∂ q^ s ∂y ;

B s = K (1 u − c ∂ q^ s ∂y + ∂ q^ s ∂q) ;

$C_s = K \left(\frac{1}{u + c} \frac{\partial \hat{q}_s}{\partial y} + \frac{\partial \hat{q}_s}{\partial q} \right).$

Figure 1 shows a comparison between the resulting matrix $|A|$ using the numerical and the approximate DOT against the value of the Froude number of the flow, given a set of sediment's characteristics. In particular, we compute the norm L_1, L_2 and L_∞ of the error committed on the computation of $|A|$:

From this analysis it is clear that the approximate DOT cannot be used as a black-box: the quality of the approximation depends on the parameter K and on the Froude number $Fr = u/c$. Furthermore, for any value of K, the approximation of $|A|$ fails for near-critical flow condition, i.e. $0.8 \le Fr \le 1.2$.

2.4 Hyperbolicity of the system of equation

In the framework of morphodynamics, the hyperbolicity of the system (1) was proved to be dependent on the chosen sediment load transport formulations (7). However, for a general form of q_s like the one shown in equation (8), the domain of hyperbolicity of the system is very large (i.e., $|Fr| < 6$ (7)), so almost all physical problems belong to this theoretical domain.

For the sake of simplicity, in the tests of the model, a simplified formulation of equation (8) due to Grass (12) is used, in which θ_c is zero and the other physical parameters collapse into an empirical coefficient.

Figure 1. Norms L 1 , L 2 and L ∞ of the error (17) committed on the computation of |A|using the approximate DOT instead of the numerical one, in the case of $K = 3.5 \cdot 10^{-5}$. The three norms are plotted against the Froude number Fr= u/c. Table 1. L 1 , L 2 and L ∞ norms of the pointwise error that the numerical model committed at time t = 7.0 [s] with respect to the reference solution of Figure 2, using the analytical eigensystem of section 2.2 and its approximation proposed in section 2.3. Analytical |A| Approximate |A| L 1 L 2 L ∞ L 1 L 2 L ∞ y 1.0e-5 1.1e-5 3.2e-5 1.0e-5 1.1e-5 3.3e-5 q 1.7e-6 1.8e-6 5.5e-6 2.3e-6 2.4e-6 4.7e-6 z 4.0e-6 1.0e-5 7.6e-5 3.9e-6 1.1e-5 7.7e-5 Thus, from now on, the sediment transport is described by equation (18): where the constant A_g depends on grain properties (e.g., $A_g = 0.005$ [s^2/m]); while the usual value of the exponent is $m_g = 3$. 3 NUMERICAL TEST CASES In order to prove the validity of the P 0 P 2 -ADER model we performed several test cases: we reproduced an analytical smooth solution (2); we studied some Riemann Problems (e.g., an original reference solution or the Test a presented in (16)); we verified that the model can reproduce exactly the situation of the lake at rest (i.e., it is well balanced); we checked that the equation are correctly coupled following the approach of (7). All these tests were well reproduced by the P 0 P 2 pathconservative scheme with the numerical and analytical computation of |A|; otherwise, using its approximation the model has some limitation as we expect from the results shows in Figure 1. To save space, in this section we can present only three test cases.

Figure 2. Reference solution described by equation (19),

proposed by Berthon et al. (2), with q = 2 [$m^3/(s \cdot m)$],

$A_g = \alpha = \beta = 0.005$ and Cost = 1, computed at time

t = 7.0 [s].

3.1 Test 1: a smooth analytical solution

The first reference solution is a smooth analytical solu

tion of the steady state condition of the water flow,

proposed by Berthon et al. (2). According to them,

for a given uniform discharge q and for a sediment

load transport as in (18), the analytical solution of equation (1) is:

in which α, β, Const are constants.

Figure 2 shows the reference solution of the system of equations (19) with $q = 2$ [$m^3/(s \cdot m)$], $A_g = \alpha = \beta = 0.005$ and Const = 1. Furthermore, setting $\xi = 1$ and $d_m = 3.1 \cdot 10^{-4}$ [m], the resulting value of the small parameter K is:

with $q_0 = U_0 Y_0 = q$ and $Y_0 = \sqrt[3]{q^2/g}$.

Table 1 reports the L_1, L_2 and L_∞ norms of the pointwise errors that the numerical model committed respect to the reference solution of Figure 2. In particular, we discretize the domain with 100 cells and compute the Osher jumps of equation (4) with the analytical eigensystem presented in (6) and its approximation proposed in this paper, section 2.3. For the sake of brevity, we omitted the results obtained using the numerical DOT Riemann solver because they are exactly the same of the one obtained with the analytical DOT formulation.

The results of Table 1 shows that, for this smooth solution, the numerical P_0P_2 model well fits with the reference solution and it is not influenced from the choice of the method used to compute $\| A(\Psi(s_j)) \|$. Thus, the method proposed in section 2.3 well approximates the analytical formulation of Λ, R and R^{-1}. 3.2

Test 2: a movable-bed Riemann Problem The second reference solution that we considered is an analytical solution of a Riemann Problem with movable bed. The used initial conditions in double precision are: for the left and right states, respectively. To perform this test we used again the Grass bed load transport (18) with $A_g = 0.005$ and $m_g = 3$. Thus, setting $\xi = 1$ and $d_m = 3.1 \cdot 10^{-4}$ [m], equation (20) returns that $K = 2.3 \cdot 10^{-3}$ being $q_0 = 1$ [$m^3/(s{\cdot}m)$] and $Y_0 = \sqrt[3]{q_0^2/g}$. The spatial domain is again discretized by 100 cells. Figure 3 shows the overlap of the analytical solution and the numerical simulations with the three considered methods. This figure shows that the three numerical simulations are very close to the reference exact solution of the Riemann problem associated to this movable bed Riemann Problem. Looking at the norms of the pointwise error shown in Table 2, it is evident that, if the Froude number is small enough, there are not appreciable differences between the analytical DOT and the approximated one. In other words, all the norms shown in Table 2 are of the same order of magnitude for both the considered methods. Nevertheless, the fact that all the error measures presented in Table 2 are bigger than the ones of Table 1 means that this second test case is more severe for the whole model in general, specially for the presence of the central shock wave. Again, we omitted the results obtained using the numerical DOT Riemann solver because they are exactly the same obtained with the analytical DOT formulation. 3.3 Test 3: an analytical linearized solution of near-critical flow conditions The last test case presented here is an extension of the analytical linearized solution proposed by Lyn and Altinakar (14) and already tested in (3). Because of the near-critical condition of the water flow (i.e., Fr= 0.9), the approximate DOT Riemann solver can't be used in this test case. Thus, the main task of this reference solution is to give us the possibility of validate the whole numerical model. Following the approach of (3), system (1) can be linearized as follows:

Figure 3. Analytical and numerical water level and bottom topography for the movable-bed dam break of initial

conditions (21), at time $t = 1.5$ [s]. The Grass bed load transport (18) is used, with $A_g = 0.005$ and $m_g = 3$.

Table 2. L_1, L_2 and L_∞ norms of the pointwise error that

the numerical model committed at the end of the simulation

shows in Figure 3, using the analytical eigensystem of section

2.2 and its approximation proposed in section 2.3.

	Analytical $\|A\|$			Approximate $\|A\|$		
	L_1	L_2	L_∞	L_1	L_2	L_∞
y	3.8e-3	6.4e-3	2.4e-2	3.8e-3	6.4e-3	2.4e-2
q	1.7e-2	4.1e-2	2.4e-1	1.7e-2	4.1e-2	2.4e-1
z	1.4e-3	2.0e-3	1.2e-2	1.4e-3	2.0e-3	1.2e-2

with:

where subscript U indicates the uniform unperturbed

state $w_U = \{y_U, q_U, z_U\}^T$, u_U is the uniform velocity,

Fr_U is the Froude number of the unperturbed state and

φ_U is a parameter related to sediment transport by the

following expression:

in which a_g and b_g have the same meaning of the

parameters A_g and m_g of the Grass formula (18), but

operatively they take different values for the presence

of the threshold velocity u_{cr}.

Once φ_U is given, the linearized system (22) can be

analytically solved by adopting the classical character

istic method, computing numerically the eigensystem

of (23). However, it is not possible to fix the parameter

φ_U into the numerical model. Thus, to have the same

load transport discharge in (1) and in (22), one may

have to set the value of u_{cr} into the numerical model, but using equation (18) it is also not possible to set u_{cr}. So, to avoid this problem, it is necessary to rewrite equation (24) in order to solve the equation $u_{cr} = 0$ in terms of b_g and, finally, set $A_g = a_g$ and $m_g = b_g$. In this way we found: being $\varphi_U = 2.5 \cdot 10^{-3}$, $a_g = A_g =$

$3.4 \cdot 10^{-4}$, the uniform flow depth $y_U = 1$ [m] and the uniform flow velocity: For this reference solution, the initial condition for the numerical simulation are: For the simulations, we discretized the spatial domain with 100 cells and we imposed the simple periodic boundary condition at the boundary of the spatial domain. The results of the simulations with analytical and numerical DOT Riemann solver are shown in Table 3 and Figure 4. Both of them confirm that the numerical model well reproduces the two small waves arising from the initial bottom topography. Furthermore, looking atTable 3.3, it is clear that the results coincide using the two method considered here. For the three test cases presented in this section, but also for several other test cases not shown here for space reasons, we can conclude that the numerical model works fine and it is a good test bench. Moreover,

Table 3. L_1, L_2 and L_∞ norms of the pointwise error that the numerical model committed at the end of the simulation shows in Figure 4, using analytical and numerical eigensys tem.The results obtained using numerical and analytical DOT Riemann solver coincide.

	Analytical $\|A\|$			Numerical $\|A\|$		
	L_1	L_2	L_∞	L_1	L_2	L_∞
y	8.9e-7	2.0e-6	6.7e-6	8.9e-7	2.0e-6	6.7e-6
q	2.1e-8	3.7e-8	1.2e-7	2.1e-8	3.7e-8	1.2e-7
z	1.0e-2	2.4e-2	8.5e-2	1.0e-2	2.4e-2	8.5e-2

Figure 4. Numerical test with analytical and numerical DOT Riemann solver upon the linearized analytical solution pro posed by Lyn & Altinakar (14), at t = 20 [s]. We omitted the water discharge because the result are analogous at what it is shown here for the water depth and the bottom topography.

being the structure of the code exactly the same for the three methods, the differences in the final results depends only on the used Riemann solver: if the Rie mann solver return the same Osher jump terms, the global results must be exactly the same, as this last test case shows.

4 COMPUTATIONAL COSTS ANALYSIS

The good results presented in section 3 allow us to push over the comparison between the three different Rie mann solver. In particular the computational cost of each method is evaluated, and a measure of the advan tage of the using the analytical formulations for $|A|$ instead of the numerical one is given.

Figure 5 shows the run durations for the Test 1 of section 3.1 on varying the number of cells that are used to discretize the space domain. These simula tions were carried out with a single core of an Intel Core i7 at 3.1 GHz. The numerical model is written in Fortran-95 and it is built by a ifort compiler that uses the lapack95 package of the Intel Math Kernel Library to compute numerically the eigensystem of a matrix. Figure 5. Computational costs analysis, i.e. durations of the runs with the three DOT Riemann solvers studied on varying the number of cells used to discretize the space domain in order to solve Test 1 (section 3.1). From the mathematical point of view, we integrated equation (3) adopting the Gauss Lobatto G-point quadrature rule with three G-point. Thus, according to equation (4), for each cell interface we computed the matrix $|A|$ three times. Looking at Figure 5, it is clear that both analytical and approximate methods are faster than numerical DOT: in particular, for more than 200 cells, they are three times faster than numerical DOT. We notice that the increment of efficiency is strictly related to the number of G-point choice, that means that the chosen of the quadrature rule has an important impact on the duration of a run. Moreover, the computational cost of the numerical eigensystem should not be underestimated when a new code is developed, because, as Figure 5 shows, its total computational cost can grow very quickly with the number of interfaces between cells; specially for 2D or 3D models. One last consideration can be done comparing the analytical and numerical DOT: Figure 5 shows that the approximate method is a little bit faster than the analytical one. Quantitatively, the saving of time results to be between 5% and 10%. However, because of the limitation due to the approximation, the advantage can be real only if: the spatial domain is very large, the simulation time is very long and the water flow is characterized from Fr� 1. 5 CONCLUSION A P 0 P 2 -ADER numerical model was built as a test bench for three different implementation of the DOT Riemann solver for the Shallow Water Equations coupled with the Exner sediment balance equation. These methods approximate the fluxes of the finite volume scheme with the Osher jumps term using the path conservative method. In particular they calculate the second part of equation (3) computing in different ways the eigensystem of the Jacobian matrix A(W). We defined numerical DOT the method that uses a numerical routine; analytical DOT the method that implements the analytical formulation proposed by (6)

written in the non-dimensional form proposed in sec

tion 2.2; approximate DOT the method that uses an

original approximation of the eigensystem of A(W)

for the case of Fr� 1. From the comparison between

these different implementations, we obtained that the

analytical DOT produce exactly the same results of the

numerical DOT, but this latter method is much more

resource demanding than the former. Otherwise, the

approximate DOT is the most efficient method of the

three that we studied, but the limitations due to approx

imations must be considered for general problems.

Thus, unless in case of very big spatial domain or very long time to simulate and water flow characterized by

Fr▮ 1, the use of analytical DOT for SWE-Exner

model is recommended. However, the approximate

formulation of the eigensystem is much more easy than

the analytical one, thus this new formulation can rep

resent a starting point for further developments (e.g.,

the role of the gravity on the bottom topography and

on the stability of dune’s shape in sandy riverbeds).

[1] Berthon, C., Boutin, B., and Turpault, R. Shock profiles for the shallow-water Exner models.Advances in Applied Mathematics and Mechanics 7, 03 (2015), 267–294.

[2] Berthon, C., Cordier, S., Delestre, O., and Le, M. H.An analytical solution of the shallow water system coupled to the Exner equation. Comptes Rendus Mathematique 350, 3-4 (2012), 183–186.

[3] Caleffi,V.,Valiani,A., andBernini,A.High-order balanced CWENO scheme for movable bed shallow water equations. Advances in Water Resources 30, 4 (2007), 730–741.

[4] Caleffi, V., Valiani, A., and Li, G. A comparison between bottom-discontinuity numerical treatments in the DG framework. Applied Mathematical Modelling (oct 2015), in press.

[5] Castro, M. J., LeFloch, P. G., Muñoz-Ruiz, M. L., and Parés, C. Why many theories of shock waves are necessary: Convergence error in formally pathconsistent schemes. Journal of Computational Physics 227, 17 (sep 2008), 8107–8129.

[6] Castro Díaz, M., Fernández-Nieto, E., and Ferreiro, A. Sediment transport models in Shallow Water equations and numerical approach by high order finite volume methods. Computers & Fluids 37, 3 (mar 2008), 299–316.

[7] Cordier, S., Le, M. H., and Morales de Luna, T. Bedload transport in shallow water models: Why splitting (may) fail, how hyperbolicity (can) help.Advances in Water Resources 34, 8 (2011), 980–989. [8] Dumbser, M., Balsara,

D. S., Toro, E. F., and Munz, C. D. A unified framework for the construction of one-step finite volume and discontinuous Galerkin schemes on unstructured meshes. Journal of Computational Physics 227, 18 (2008), 8209–8253. [9] Dumbser, M., and Käser, M. Arbitrary high order non-oscillatory finite volume schemes on unstructured meshes for linear hyperbolic systems. Journal of Computational Physics 221, 2 (2007), 693–723. [10] Dumbser, M., and Toro, E. F.A Simple Extension of the Osher Riemann Solver to Non-conservative Hyperbolic Systems. Journal of Scientific Computing 48, 1-3 (2011), 70–88. [11] Exner, F. M. Über die Wechselwirkung zwischen Wasser und Geschiebe in Flüssen. Akademie der Wissenschaften, Sitzungsberichte 134, Wien, Austria, 1925. [12] Grass, A. J. Sediment transport by waves and currents. University College, London, Dept. of Civil Engineering, 1981. [13] Hou, T. Y., and LeFloch, P. G.Why nonconservative schemes converge to wrong solutions: error analysis. Mathematics of Computation 62, 206 (1994), 497–497. [14] Lyn, D. a., and Altinakar, M. St. Venant–Exner Equations for Near-Critical and Transcritical Flows. Journal of Hydraulic Engineering 128, 6 (2002), 579–587. [15] Meyer-Peter, E., andMüller,R.Formulas for BedLoad Transport. Proceedings of the 2nd Meeting of the International Association of Hydraulic Research (1948), 39–64. [16] Murillo, J., and García-Navarro, P. An Exnerbased coupled model for two-dimensional transient flow over erodible bed. Journal of Computational Physics 229, 23 (2010), 8704–8732. [17] Nord, G., and Esteves, M. PSEM_2D: A physically based model of erosion processes at the plot scale. Water Resources Research 41 (2005), 1–14. [18] Parés, C. Numerical methods for nonconservative hyperbolic systems: a teoretical framework. SIAM Journal on Numerical Analysis 44, 1 (2006), 300–321. [19] Toro, E. F., Millington, R. C., and Nejad, L. A. M. Towards Very High Order Godunov Schemes. In Godunov Methods, E. F. Toro, Ed. Springer US, Boston, MA, 2001, pp. 907–940. [20] Vázquez-Cendón, M. E. Improved Treatment of Source Terms in Upwind Schemes for the Shallow Water Equations in Channels with Irregular Geometry. Journal of Computational Physics 148, 2 (1999), 497–526. [21] Wong, M., and Parker, G. Reanalysis and Correction of Bed-Load Relation of Meyer-Peter and Müller Using Their Own Database. Journal of Hydraulic Engineering 132, 11 (nov 2006), 1159–1168.
Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

A new Osher Riemann solver for shallow water flow over fixed or mobile bed

D. Zugliani & G. Rosatti

Department of Civil, Environmental and Mechanical Engineering, University of Trento, Trento, Italy

ABSTRACT: The Osher solver (Osher & Solomon 1982) is a well-known numerical approach to estimate

solutions of Riemann problems deriving from the finite volume method with Godunov fluxes applied to hyper

bolic systems of Partial Differential Equations (PDEs). Recently, in Dumbser & Toro (2011b), the applicability

of the solver has been extended to purely nonconservative systems. Nevertheless, shallow flows are described by

a system where both conservative and non-conservative terms are present simultaneously. Some effort must be

done to use the solver in this joined situation. In this work, we combined the conservative and non-conservative

formulation ending up with a simple but powerful extension of the Osher solver suitable for the Shallow Water

(SW) partially nonconservative PDEs systems. We also introduced a linear path in terms of primitive variables,

instead of conserved ones. This approach reduces a little bit the computational cost in cases with simple lin

ear relations between conserved and primitive variables (fixed-bed flows), while the cost reduction becomes

more important when the relation is highly nonlinerar (mobile-bed flows) and the Jacobian of the fluxes can be

expressed only in term of primitive variables. Finally, we exploited the possibility to use an explicit expression

of the path integral of the non-conservative terms instead of a numerical approximation of it, e.g. the work of

Rosatti & Begnudelli (2010).

1 INTRODUCTION

The 1D Shallow Water (SW) equations are the starting point for the description of several envi ronmental phenomena like flash floods, hypercon centrated flow, debris flow, etc. The Partial Dif ferential Equations (PDEs) system is characterized by the presence of both conservative and noncon servative terms. Due to the hyperbolic nature of the PDEs system, a good numerical strategy for its integration is the finite volume method with Godunov type flux evaluation. Among many approx imated Riemann solver present in the literature, a particular one is the Osher solver developed by Osher & Solomon (1982) and revisited by Dumbser & Toro (2011a) for the conservative and by Dumbser & Toro (2011b) for the nonconservative system from an ‘universal’prospective (see section 2). In this paper (section 3) we introduce a new universal Osher solver that is a natural extension of original one. The numer ical solver is applied (section 4) to the shallow water equation with discontinuous bed and to the two-phase isokinetic mobile bed equation.

2 EXISTING OSHER SOLVER

A generic one dimensional hyperbolic system com posed by m partial differential equations (PDEs) conservation laws can be written in compact form as where U and F (U) are the conserved variables and the conservative fluxes vector expressed in term of U respectively. The quasi-linear form of this system is where

A U (U) is the Jacobian matrix of the conservative fluxes respect to the conserved variables The numerical solution of equation (1) using the finite volume method reads where i states the cell i▯x, n is the time n▯t, U is the vector of the cell average conserved variables and F is the vector of the fluxes. The evaluation of the fluxes could be done using different approximated Riemann solver. One of them is the Osher solver developed by

Osher & Solomon (1982) and it is based on the the

assumption that it is possible to split the flux in a

positive part and a negative one

where

Given the left value U L and the right one U R of any

Riemann problem, the Osher flux is evaluated using

the following expression

With some mathematical manipulation, it can be

rewritten as

where F L and F R are the fluxes evaluated using the left

and the right conserved variables respectively, while

and R is the matrix where the columns are the right

eigenvectors of A U , R −1 is its inverse and is the

diagonal matrix of the eigenvalues.

In the expression (8) appears, as first term, the cen

tral part of the fluxes, while the integral represents the

so called numerical viscosity.

Following Osher & Solomon (1982) it is neces

sary to know all the intermediate states of the solution

(e.g. using rarefaction fan), in order to been able to

solve the integral (8) in an explicit form. To overcome this difficulty, Dumbser & Toro (2011a) developed

a new methodology by using an integral path in the

phase-space. With this strategy, equation (8) can be

rewritten as

where (s) is the path that links the left state U L with

the right oneU R in the phase-space.� (s) is a Lipschitz

continuous function defined in the interval s ∊ [0, 1]

with (0) =U L and (1) =U R . The choice of the

path is now independent of the intermediate states,

making the solver easier to be implemented and used.

Several hyperbolic problems, however, are described

by fully non-conservative systems. This type of sys

tems can be written in vectorial form as

where H (U) is the matrix of the non-conservative

fluxes. These systems are formally different from the conservative ones (2) since H (U) is not the Jacobian of the fluxes. The solutions of these problems are quite different from the previous ones and need particular care in the development of a Riemann solver. The extension of the Osher solver (10) to the non-conservative systems makes use of the theory of Dal Maso, LeFloch, & Murat (1995) on the nonconservative products. Following this theory the compatibility condition on the fluxes (for shake of clarity the fluxes for these type systems is represented with G) can be written as and, from this definition, Dumbser & Toro (2011b) defined the fluxes for the Osher solver for fully nonconservative systems in the following way The numerical viscosity term that appears in expressions (10), (13) and (14) can be evaluated, following Dumbser & Toro (2011b), using the simplest path that connect the left and the right state. The proposed path is a straight-line segment defined as This path is used in many papers, among which we can cite Castro Díaz et al. (2013) where they say: “[...] when there are no clear indication about the correct family of paths to be chosen, the family of straight segments is a sensible choice […]". 3 THROUGH A NEW OSHER SOLVER The SW systems could not been correctly written by fully conservative (2) or non-conservative (11) PDEs, but they are described by equations composed with a mix of conservative and non-conservative terms. These mixed hyperbolic systems are written, in a quasi-liner form, as where Equation (16) is similar to the quasi-linear form (2) of the conservative system but, in the present case, the matrix A(U) contains both the Jacobian of the conservative fluxes (as for the conservative case) and also the

non-conservative fluxes (as for the non-conservative

case). The finite volume method applied to this type

of hyperbolic systems reads

where, due to the non-conservative characteristics of

some terms, the outgoing fluxes from the cell i (i.e.

F −

i+ 1 2) are different from the incoming ones in the cell

i + 1 (i.e. F + i+ 1 2).

In order to evaluate these fluxes, we assumes, in a

similar way as for the original Osher solver, the exis

tence of a splitting between the conservative fluxes and

the non-conservative ones. In this way it is possible to

write the following relation

where F i+ 1 2 is described by equation (10) and D ± i+ 1 2

with the equations (13) and (14). With this statement

we can write

that is the New Osher solver in Conservative variables

(NOC) for a generic hyperbolic system containing con

servative and non-conservative terms. From now on,

for shake of clarity, we neglect the dependency of in the matrices.

We can easily check that for a conservative sys

tem (i.e. H= 0 and | A U | = | A U |), the solver reduces

exactly the conservative one (10). Also the compati

bility condition that reads

as for the non-conservative system could be easily

proofed.

3.1 New Osher solver in primitive variables

This new Osher solver in conservative variables is

developed for an hyperbolic system of PDEs where

the fluxes could be written using the conservative

variables. An example is the well known Shallow

Water Equations with discontinuous bed (SWE) ana

lyzed later on. Nevertheless, writing the fluxes for an

hyperbolic system in term of conservative variables

is not always possible. Indeed, one example is the

system of PDEs describing the two-phase isokinetic

flow over mobile bed (see section 4.2). However is possible to rewrite these type of systems in compact form as where U and W are the vectors of the conserved and primitive variables respectively, while F (W) and H (W) are the conservative and non-conservative fluxes expressed in term of primitive variables. In quasi linear form, this type of PDEs system reads where and is the Jacobian matrix of the conservative fluxes respect to the primitive variables, while is the inverse of the Jacobian of the conserved variables respect to the primitive ones. Comparing system (16) and (23) we notice that they are similar except for the definition of the matrix A, so the key idea is to use the NOC solver (20) changing the matrices in the two integrals.This lead to the following expression for the fluxes This simple extension has the disadvantage that the path integration is defined in conserved variables, while the matrices A W , B and H depends on the primitive ones therefore, for the evaluation of the two integrals, a change of variable from the reconstructed conserved variables to the reconstructed primitive ones is necessary. For many systems, e.g. the mobile bed system, the passage from conserved to primitive variables is quite complicated and leads to a non-liner system that must be solved several time during the integration process since, as we explained later on in section 3.3, it is done in a numerical way. Our idea is to change the path integration from the conserved variables to the primitive ones in order to overcome the complexity related to the solution of the non-linear system. Omitting all the details for the

derivations, we end up with a New Osher solver with

Primitive variable (NOP) that reads

where P (s;W L ,W R) is the path in primitive variable

connecting the left values W L and the right ones W R .

Since the path in primitive variables must have all the

property of the path in conserved variables and fol

lowing the approach used in Dumbser & Toro (2011b)

it is possible to use the segment linear path also in the

NOP in order to maintain the generality of the scheme.

This linear path in primitive variables reads

and thanks to it we can write the general structure for

the NOP

where the first term is the central part of the fluxes,

the first integral represents the so called numerical

viscosity that derives from the conservative and non

conservative fluxes, while the second integral is the

integral value of the non-conservative fluxes across

the interface of the computational cell.

3.2 Discretization of the non-conservative terms

For the free surface flow, the non-conservative terms has an important role in the solution of the Riemann problems, so particular attention must be paid for its discretization. The physical meaning of the integral of the non-conservative term, i.e. the second integral in equation (28), can be analyzed referring to the theory developed by Dal Maso et al. (1995) about the non conservative product. Following this theory, the weak solution of (23) across a discontinuity must satisfy

where S S is the speed of the traveling discontinuity. Using the definition (25) and (26) it is possible to rewrite this expression as

where last term of the equation is the integral we are

referring to, while the other terms are the classical Rankine Hugoniot (RH) condition. As proved in Rosatti & Begnudelli (2010) for the standing contact wave of the SWE with discontinuous bed and in Rosatti & Fraccarollo (2006) for the shock waves of the two phase mobile bed, the only relation valid for the description of this type of waves is the Generalized Rankine Hugoniot (GRH) conditions that reads where F L and F R is the conservative fluxes on the left and right of the wave, while D represents the pressure exerted by the fluid on the bottom step. Comparing equation (33) with (32) we notice that, for the free surface flow, the integral of the nonconservative term is nothing more than the pressure exerted by the fluid on the bed step so, in the integration of the non-conservative term in the NOP solver (30) it is necessary to use the correct path in order to satisfy equation (34). Since a general expression for the path that corroborates this equation is not easy to find or is quite complicated to deal with (e.g. the path proposed by Cozzolino et al. (2011)) our cutting-edge idea is the introduction in the NOP solver of the exact integral of the non-conservative terms so equation (34). Since we want to maintain also a part of the universality of the solver, we can use the linear path in the numerical viscosity term since no constrains on its are provided. The final version of the NOP, proposed in this article, is therefore 3.3 Numerical discretization In the final form of the proposed NOP solver (35), the numerical viscosity term must be evaluated via the solution of the integral itself. Since a closed form of this integral is not always available, it is possible to solve it using a numerical method (e.g. midpint rule, trapezoidal rule, ...). Among these methods, a good accuracy can be obtained using the Gauss-Legendre (GL) quadrature rule with three points. Following this method the integral is replaced with a summation and the resulting NOP solver is

where s_i is the position and ω_i is the weight. For the three point GL rule in the integration domain $s \in [0; 1]$, the positions and weights read

The use of GL rule for the integral of the numerical viscosity produce a simple, powerful and physically based Osher Riemann solver where the only information needed are the eigenstructure of the hyperbolic problem.

4 TEST CASES

In this section we present two applications of the new Osher solver (both NOC and NOP) presented in this paper. In particular we apply the Riemann solver to the SWE with discontinuous bed in Section 4.1, while in Section 4.2 we solve the two-phase isokinetic mobile bed system.

4.1 Fixed bed system

The homogeneous part of the classical SWE with discontinuous bed is where h is the water depth, u is the water velocity, g in the constant gravity acceleration, z b is the bed elevation, x is the longitudinal direction and t is the time as sketched in figure 1. For all the details on this type of system, we refer to the works of LeFloch & Thanh (2007) and Rosatti & Begnudelli (2010).

The test case proposed is the solution of a Riemann problem where the left state is h L = 3.0m, u L = 0.5m/s and z bL = 0.0m, while the right state is define by h R = 2.0m, u R = 2.07m/s and z bR = 0.5m. The domain is described with 250 cells, the final integration time step is 1.0s and the Courant number is set equal to 0.9.

In figure 2 is represented the comparison between analytical solution (in black solid line) and the numer ical result obtained using the NOP solver without (in red dots) and with (in blue dots) corrected non conservative term. At a first glance the two numerical solutions agree well with the analytical one. How ever, zooming in at the free surface, near the bed discontinuity (figure 3), we highlight that only the Osher solver with the correct discretizatio of the non conservative terms reproduces in a satisfactory way the exact solution. Figure 1. Sketch of the variables for the fixed bed case. Figure 2. Comparison between analytic solution (black line) and numerical solution with NOP solver without (red dots) and with (blue dots) correction on the non-conservative terms discretization for the fixed bed Riemann problem test case. In the picture are represented the free surface η= h + z b and the bottom elevation z b . Figure 3. Zoom of the free surface elevation near the bed step for the fixed bed Riemann problem test case. Black line is the exact solution while red dots and blue dots represents the NOP solver without and with correct discretization for the non-conservative term. 4.2 Mobile bed system The system of PDEs describing the two-phase isokinetic flow over mobile bed (see Armanini et al. (2009), Cao et al. (2006) and Murillo & García-Navarro (2010) for all the details on this type of system) is

Figure 4. Sketch of the variables for the mobile bed case.

where h is the flow depth, z b is the bottom elevation,

u is the velocity, is the constant submerged relative

density for the solid phase, g in the constant gravity

acceleration, x is the longitudinal direction, t is the

time, c is the concentration of the solid phase in the

flow and c b is the constant bottom concentration as

sketched in figure 4.

In order to solve this PDEs system it is necessary

to introduce an algebraic closure formulation relating

the concentration with the local hydrodynamic values,

like

Many formulations can be found in the literature (we

refer to the work of Wu (2007) for a possible list of

them) and here we use the simple relation proposed by

Armanini et al. (2009)

where β is a constant dimensionless transport param

eter.

The test case proposed is the solution of a Riemann problem where the left state is $h_L = 3.0$m, $u_L = 0.0$m/s and $z_{bL} = 0.0$m, while the right state is define by $h_R = 0.770$m, $u_R = 0.665$m/s and $z_{bR} = 1.06$m. The other parameter used are $\beta = 1.0$, $c_b = 0.65$ and �= 1.65. The domain is described with 250 cells, the final integration time step is 1.0s and the Courant number is set equal to 0.9.

In figures 5 and 6 are represented the comparison between analytical solution (in black solid line) and the numerical results obtained using the NOP solver without (in red dots) and with (in blue dots) corrected non-conservative term. At a first glance the two numerical solutions agree well with the analytical one. However, zooming in at the free surface near the shock wave (figure 7), we highlight that the Osher solver with the correct discretizatio of the non-conservative terms reproduces in a better way the exact solution.

We perform lots of other simulation both with the new Osher solver in primitive variables and the one with the conserved variables in order to evaluate their time consumption. With a systematic compari

son between the two solvers, we highlight a speed up Figure 5. Comparison between analytic solution (black line) and numerical solution with NOP solver without (red dots) and with (blue dots) correction on the non-conservative terms discretization for the mobile bed Riemann problem test case. In the picture are represented the free surface $\eta = h + z_b$ and the bottom elevation z_b . Figure 6. Comparison between analytic solution (black line) and numerical solution with NOP solver without (red dots) and with (blue dots) correction on the non-conservative terms discretization for the mobile bed Riemann problem test case. In the picture is represented the concentration c. Figure 7. Zoom of the free surface elevation for the mobile bed Riemann problem test case near the shock wave. Black line is the exact solution while red dots and blue dots represents the NOP solver without and with correct discretization for the non-conservative term. of about 3.3% for the NOP respect to the NOC. This is an advantage for the NOP solver respect to the NOC one, since saving 3.3% of time in a simulation of a realistic free surface flow event, where the timing is of the order of hours, allows to obtain accurate solutions a little bit faster. 5 CONCLUSIONS With this work, we developed a powerful but simple extension of the Osher solver for partially nonconservative PDEs systems, such as the systems describing free surface flow, using a linear path integration for

the numerical viscosity and an exact integration for

the non-conservative terms. The path integration is

performed using both conserved and primitive vari

able, and the two obtained solvers (respectively NOC

and NOP) are applied to the well known SWE with

bed discontinuity and two-phase isokinetic mobile bed

equations. With some test cases we have highlighted

that only using the corrected discretization of the non

conservative term, the numerical solution have a good

agreement with the analytical one. We also highlight

that the NOP solver is 3.3% faster than the NOC one

when the Jacobian of the fluxes can be expressed only

in term of primitive variables.

The extension to a 2D model of the new Osher solver

will be done in a forthcoming works.

LIST OF SYMBOLS

c concentration

c_b bottom concentration

g gravity acceleration

h water depth

i cell index

n time index

s integration variable

t time

u flow velocity

x space variable

z_b bottom elevation

A_X Jacobian matrix of the fluxes respect to the generic vector X

B Jacobian matrix of the conserved variables respect to the primitive ones

D pressure exerted by the fluid on the bed step

F vector of the fluxes

G vector of the fluxes for non-conservative case

H matrix of non-conservative fluxes

S_s shock speed

U vector of conserved variables

W vector of primitive variables

β dimensionless transport parameter

η free surface elevation

ω Gauss-Legendre weight submerged relative density for solid phase

t time step integration

x space discretization diagonal matrix of eigenvalues path in conserved variables

P path in primitive variables

R matrix of the right eigenvectors

X L,R generic vector X evaluated on the left or right of a discontinuity

Grid coarsening and uncertainty in 2D hydrodynamic modelling

V. Bellos & G. Tsakiris

Laboratory of Reclamation Works and Water Resources Management & Centre for the Assessment of Natural

Hazards and Proactive Planning, School of Rural and Surveying Engineering, National Technical University of

Athens, Greece

ABSTRACT: The aim of the paper is to study the impact of grid coarsening on the water depths produced by

2D hydrodynamic simulation for both steady and dynamic evolution states. For the numerical simulation, the

hydrodynamic model called FLOW-R2D is used. Initially, the model is tested against the uncertainty introduced

to the numerical results in a steady state scenario (hydraulic jump). Extending the uncertainty analysis to the

dynamic state, an experimental test is employed in which the flood wave, created by a dam break, is propagated

through a wide channel with a horizontal bed. In both of these scenaria, three grid sizes are generated and

compared. For the uncertainty quantification, a grid convergence study is performed, using the Grid Convergence

Index. It is concluded that according to the grid convergence testing of the model in the steady state, there is no

significant impact of the cell size on the produced water depths. On the contrary, the numerical results seem to be affected from the cell size in the dynamic evolution phenomenon, as the flood wave propagation.

1 INTRODUCTION

1.1 Basic notions

The various numerical solutions which can be imple mented in each case study have as a consequence that the numerical results derived are not unique, but they fall in a range of values. This uncertainty of the numer ical results can be caused by a number of reasons, such as the type of the Partial Differential Equations (PDE) used in the simulation, the method used for the numer ical solution of the PDEs, and the grid cell size used for the representation of the computational domain.

This paper deals with the assessment of impact of the grid coarsening, on the numerical results derived from a two-dimensional (2D) hydrodynamic simula tion, in both steady and dynamic evolution states. This uncertainty is investigated by the grid convergence study and in particular with the estimator known as Grid Convergence Index (GCI). The variable tested is the water depth of the inundated area.

For the steady state scenario, the formation of a hydraulic jump in a hypothetical rectangular channel is used, whereas for the dynamic evolution scenario, the numerical simulation of an experimental bench

mark test is employed. The test presents a flood wave

created by a partial dam-break and propagated through

a horizontal dry bed with an isolated obstacle.

1.2 FLOW-R2D model

In the present study, the recently developed 2D hydro

dynamic model called FLOW-R2D, is used (Tsakiris &

Bellos 2014).This model is based on the fully dynamic form of the 2D Shallow Water Equations (2D-SWE). The numerical method used is the Finite Difference Method and a modified explicit McCormack numerical scheme (McCormack 1969), adding artificial viscosity, through a diffusion factor (ω).The scheme is implemented in a cell-centered, non-staggered computational grid and in its original form has been used also by various researchers (Liang et al. 2006; Kalita 2016). A special attention is paid to the representation of wet/dry conditions, in which a water depth threshold (h_{dry}) is inserted in the model to distinguish the wet and dry cells. Finally, it is noted that the model has been extensively tested and applied in various case studies, such as Bellos & Tsakiris 2015a, b and c. 1.3 Grid convergence study The grid convergence study is performed through the GCI, with which the discretisation error introduced to the numerical solution, due to the grid coarsening or refinement, is quantified. GCI is proposed by Roache (1993, 1994, 1998) and is based on the Richardson extrapolation. According to this extrapolation, a variable f derived from the numerical solution is calculated as follows: where $f_{L\to0}$ is the value of the exact or asymptotic solution whereas the grid cell size bounds to zero and the functions g_1, g_2 etc. are independent from the grid cell size. In a second-order accuracy method (such as the McCormack numerical scheme), the function g_1 is

zero. Deriving two values for the variable (f_1 and f_2) for

two grid cell sizes (fine and coarse, respectively) and

neglecting the Higher Order Terms (HOT), the exact solution can be written as:

where r is the grid coarsening ratio (L_2/L_1).

Extending the above procedure to the p-th order of accuracy methods, the above equation can be generally written:

The GCI is an approximation of the actual fractional error between the exact and the coarse or the fine solu tion, introduced to the numerical results due to the grid coarsening or refinement, respectively, incorporating a safety factor (F_s). For the coarse grid the GCI is calculated by the following equation:

Apart from the quantification of uncertainty, it is interesting to check whether the observed order of con vergence is equal to the formal order of convergence, which in turn is equal to the order of accuracy of the numerical method.The observed order of convergence can be defined using three values of the variable stud ied, which are derived by numerical simulations with different grid cell sizes. Assuming that the ratio of the grid coarsening is steady, the observed order of convergence (p_o) can be calculated by the following expression:

If $a \approx r^{p_o}$, then p_o is the observed order. The param eter a is the GCI ratio:

2 GRID CONVERGENCE STUDY

2.1 Steady state scenario

The first test with the FLOW-R2D model is per

formed implementing a steady state scenario, namely

a hydraulic jump formation in a rectangular channel.

More specifically, the set-up of the above scenario con

sists of a prismatic orthogonal channel 100 m long, Figure 1. Numerical results for the three grid cell sizes against the analytical solution for the steady state scenario. Figure 2. GCI values for the two steps of grid coarsening and the steady state scenario. where the bottom slope for the first half length section is 3% (steep slope/supercritical conditions) and for the rest 0.03% (mild slope/subcritical conditions). The width of the channel is 20 m, the Manning coefficient is $n = 0.030$ s/m $^{1/3}$ and the constant inflow is $Q = 10$ m ^{3} /s. Three different grids are generated for the computational domain representation with different square cell sizes: $L_1 = 0.5$ m, $L_2 = 1.0$ m and $L_3 = 2.0$ m. For the numerical simulation, the time step is determined that the Courant-Friedrichs-Lewy condition, assumed the Courant number (CFL) is 0.1.The wet/dry threshold is determined $h_{dry} = 3 \times 10^{-3}$ m and the diffusion factor $\omega = 0.99$. In Figure 1, the water depth derived from the numerical simulations for each cell size is presented along the streamwise central axis of the channel. A comparison is also made towards the analytical solution based on the Manning's equation (Bellos 2014). In Figure 2, the GCI values are plotted for each step of coarsening (from L_1 to L_2 and from L_2 to L_3) along the streamwise central axis of the channel. The safety factor is considered as $F_s = 1.25$, according to the related literature (Roache 1998). Also in Figure 3 and 4, the observed order of convergence and the ratio a/r^{p_o} are plotted along the central axis of the channel, respectively.

Figure 3. Observed order of convergence of the FLOW-R2D

model for the steady state scenario.

Figure 4. The ratio a/r^{p_o} for the steady state scenario.

2.2 Dynamic evolution scenario

In this section an attempt is made to extend the grid convergence analysis from the steady state, to the dynamic evolution state, in a complex domain. For this purpose, the water depths derived from numerical simulations of an experimental dam-break flood wave, are used. The experiment is made by Soares-Frazao & Zech (2007) and consists of a well-known benchmark test, included in the IMPACT Project (Zech & Soares Frazao 2007). Various other researchers have used this test in order to validate their models (e.g. Abderrezzak et al. 2008).

In fact, the examined flood wave is created by a partial dam break. After the dam failure, this flood wave is propagated through a horizontal dry bed, with an isolated obstacle, which is oblique to the flow direction. An indicative view of the experimental set-up is presented in Figure 5 with an extra detail in the area around the isolated obstacle. The initial water depth of the tank which represents the reservoir is 0.40 m. The water depths are recorded with respect to time at six gauges (G1-G6) which are also presented in Figure 5.

Three grids are generated for the computational domain representation with square cell sizes: L_1 = 5 cm, L_2 = 10 cm and L_3 = 20 cm. For the numerical simulation, the Courant number is determined as CFL = 0.01. The wet/dry threshold is determined

$h_{dry} = 3 \times 10^{-4}$ m, the diffusion factor ω= 0.99 and

the Manning coefficient n = 0.010 s/m $^{1/3}$ .The isolated Figure 5. Experimental set-up of the dynamic evolution scenario. Figure 6. Numerical results (water depth in m) at G1 for the three grid cell sizes against experimental results for the dynamic evolution scenario. Figure 7. Numerical results (water depth in m) at G2 for the three grid cell sizes against experimental results for the dynamic evolution scenario. obstacle, the dam and the walls of the experimental setup are represented by the modified reflection boundary technique (Bellos & Tsakiris 2015a). Figures 6 to 11 present the evolution of the water depth with respect to time recorded at the six gauges and derived for the various grid cell sizes simulations, against the experimental data. In Figures 12 to 17, the GCI values are presented with respect to time. The

Figure 8. Numerical results (water depth in m) at G3 for

the three grid cell sizes against experimental results for the

dynamic evolution scenario.

Figure 9. Numerical results (water depth in m) at G4 for

the three grid cell sizes against experimental results for the

dynamic evolution scenario.

Figure 10. Numerical results (water depth in m) at G5 for

the three grid cell sizes against experimental results for the

dynamic evolution scenario.

safety factor is considered as $F_s = 1.25$. Except from

the GCI values, an error estimator between the numer

ical and the experimental results, is also plotted. This

error estimator (SE_L) is calculated for each simulation

as follows: Figure 11. Numerical results (water depth in m) at G6 for the three grid cell sizes against experimental results for the dynamic evolution scenario. Figure 12. GCI and SE L values with respect to time for the G1. Figure 13. GCI and SE L values with respect to time for the G2. where f L is the numerical and f ex the experimental result. It should be noted that the 5 second moving average for both the GCI and the SE L values is plotted. Besides, the observed locally extreme values are excluded from the figures. In Figure 18, the calculated order of convergence is presented. Finally, Figure 19 presents the ratio a/r p o with respect to time for all the six gauges. It can be seen that the observed order of convergence

Figure 14. GCI and SE L values with respect to time for

the G3.

Figure 15. GCI and SE L values with respect to time for

the G4.

Figure 16. GCI and SE L values with respect to time for

the G5.

is plotted after the first 15 seconds due to the fact that

the ratio a/r p o is stabilised to the unit after this point in

time.

Except from the SE L , the performance of the model

against the experimental results is also investigated

correlating the numerical with the experimental

results, using the well-known Pearson coefficient

(PCC). The PCC values for the six gauges are pre

sented in Table 1. Figure 17. GCI and SE L values with respect to time for the G6. Figure 18. Observed order of convergence of the FLOW-R2D model for the six gauges and the dynamic evolution scenario. Figure 19. The ratio a/r p o for the six gauges and the dynamic evolution scenario. 3 CONCLUDING REMARKS 3.1 Remarks Regarding the convergence study of the steady state is concerned, it is concluded that the uncertainty introduced to the numerical results derived by the FLOW-R2D model due to the grid coarsening, is relatively small. The maximum GCI value is 5.5% for

Table 1. PCC values for the various numerical simulations.

Gauge	L (cm)	PCC
G1	5	0.561
G1	10	0.62
G1	20	0.458
G2	5	0.842
G2	10	0.543
G2	20	0.634
G3	5	0.807
G3	10	0.778
G3	20	0.595
G4	5	0.852
G4	10	0.803
G4	20	0.694
G5	5	0.644
G5	10	0.625
G5	20	0.511
G6	5	0.996
G6	10	0.997
G6	20	0.994

the first step of grid coarsening (from L = 5 cm to L = 10 cm) and 8.8% for the second step (from L = 10 cm to L = 20 cm) along the streamwise central axis of the channel, except from a few local negative values. From the graphical comparison between the analytical solu tion and the numerical results derived for the three grid cell sizes, the simulations can be characterised as satisfactory.

The observed order of convergence seems to devi ate from the formal one, which is 2. According to the analysis, the order of convergence ranges from −3 to 8 for the supercritical section (uniform and gradually varied flow) of the flow and is relatively stabilised after the hydraulic jump with a range from 0.8–1.2. Probably these discrepancies are created due to the dif fusion factor incorporated into the FLOW-R2D model which adds artificial viscosity (Roache 1997). How ever, similar comments have been made also by other researchers (Roy 2003).

Regarding the convergence study of the dynamic evolution scenario, it seems that the uncertainty intro duced to the numerical results in a complex compu tational domain with a dynamic evolution, is quite significant. Both of the values of GCI and SE L val ues cover a wide range, whereas a peak is observed when the flood wave reaches each gauge, despite of the fact that according the PCC values, the simulation can be characterised as satisfactory. As expected, the finest grid performs better than the coarser ones.

Since the ratio a/r p o is stabilised to the unit after the first 15 s, the observed order of convergence can be determined from this moment until the end of the experimental process. The observed order of convergence has significant discrepancies from the respective formal.

The simulations can be classified into three differ ent cases as follows: a) the G2 gauge: after the initial

Bellos, V. & Tsakiris, G. 2015b. Flash flood simulation in small catchments based on combined hydrodynamic and hydrologic approach. Proc. 9th world congress of EWRA, Istanbul, 10–13 June 2015. (e-proceedings).

Bellos, V. & Tsakiris, G. 2015c. 2D flood modelling: the case of Tous dam break. Proc. 36th world congress of IAHR, Delft-the Hague, 28 June–3 July 2015. (e-proceedings).

Kalita, H.M. 2016. A new total variation diminishing predic tor corrector approach for two-dimensional shallow water flow. Water Resources Management 30(4): 1481–1497

Liang, D., Lin, B., Falconer, R.A. 2006. Simulation of rapidly varying flow using an efficient TVD-McCormack scheme. International Journal for Numerical Methods in Fluids 53: 811–826.

McCormack, R.W. 1969. The effect of viscosity in hyper velocity impact cratering. AIAA hypervelocity impact conference, Cincinnati: Paper 69–354.

Roache, P.J. 1993. A method for uniform reporting of grid

refinement studies. Proc. 1th AIAA Computational Fluid

Dynamics conference, Orlando, 2:939–948. Roache, P.J. 1994. Perspective: a method for uniform reporting of grid refinement studies. Journal of Fluids Engineering 116(3): 405–413. Roache, P.J. 1997. Quantification of uncertainty in Computational Fluid Dynamics.Annual Review of Fluid Mechanics 29: 123–160. Roache, P.J. 1998. Verification and validation in computational science and engineering. Hermosa publishers. Roy, C.J. 2003. Grid convergence error analysis for mixedorder numerical schemes. AIAA Journal 41(4): 595–604. Soares-Frazao, S. & Zech, Y. 2007. Experimental study of dam-break flow against an isolated obstacle. Journal of Hydraulic Research 45: 27–36. Tsakiris, G. & Bellos, V. 2014. A numerical model for twodimensional flood routing in complex terrains. Water Resources Management 28(5): 1277–1291. Zech, Y. & Soares-Frazao, S. 2007. Dam-break flow experiments and real-case data. A database from the European IMPACT research. Journal of Hydraulic Research 45: 5–7.
Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Migration characteristics of a meandering river: The Madhumati river,

Bangladesh

Mohd. Sarfaraz Banda

Department of Hydraulic Engineering, Leichtweiß-Institute (LWI), TU Braunschweig, Germany

Shinji Egashira

International Centre for Water Hazard and Risk Management (ICHARM), PWRI, Japan

ABSTRACT: Linearized 1-D models are widely used to simulate morphodynamic behavior of natural mean

dering rivers. Ikeda et al. (1981) developed a linear solution to predict near-bank excess velocity based on shallow

water flow model. In their approach, bank erosion rate is linearly related to the near-bank excess velocity through

an empirical erosion coefficient. Hasegawa (1989) proposed a bank migration model, which is based on the same linear relation as Ikeda et al.'s, but includes the erosion rate formula that is derived from integration of sediment continuity equation. This study aims to investigate whether these meander migration models are able to reproduce the observed behavior of a meander bend of the Madhumati River (Bangladesh). Simulation results show that near-bank excess velocities are small in the upstream of bend apex, reach to the maximum value at bend apex and then decrease in the downstream. Correspondingly, the bank migration follows the same trend. The agreement between predicted and observed migration is good in the upstream of bend apex, but the migration is underestimated in the downstream of bend apex. On the contrary, when an influence of the near-bank excess flow depth on the bank migration is added to Hasegawa's model, it is able to simulate the downstream migration well. Although the near-bank excess velocities and observed near-bank excess flow depths were out of phase, they together amplified the bank migration rate, which resulted in shifting the migration towards the downstream direction. However, this method necessitates careful estimation of flow width from asymmetrical cross-sections at bend.

1 INTRODUCTION

River bank erosion has been identified as a type of disaster in Bangladesh. Throughout the country about 1200 km of riverbanks are under severe erosion problem. Around eight thousand hectares of lands are eroded every year. The impact of land loss involves first and foremost the loss of homestead land, housing structures, crops, trees and households. Loss of home steads compels people to move to new places without any option and thus puts them in disastrous situation. Difficulties in river bank erosion control in Bangladesh arise both from economical or financial and technological reasons. Economic and financial constraints appear due to very low return from the investment. This implies that only the priority sites can be brought under structural measures. However the location and extent of bank erosion is not yet pre dictable in reasonable time ahead. Some difficulty for evaluating bank erosion hinders the proper planning for river management.

Prediction of channel shifting using numerical sim ulations has been identified as an important tool to conduct site adaptive countermeasures in river management. 1-D models are widely used in the simulation of short and long term migration of meandering rivers. These models study the longitudinal profiles of the cross-section averaged properties of flow and sediment transport in rivers (Howard, 1992; Crosato, 2008). Present research is aimed to test the possibility of using a mathematical model, based on river mechanics principles, as a tool to assess the migration characteristics of meandering rivers. The meander migration models proposed by Ikeda et al. (1981) and by Hasegawa (1989) were applied to predict the migration of a meander bend of the Madhumati River and were examined by comparing the simulated results and corresponding field data. It is found that the both model could be acceptable for evaluating the channel shift although some improvements may be necessary. Especially, Hasegawa’s model will predict well the bank erosion and corresponding channel shift if the flow width is specified carefully. Nevertheless, such meander migration models provide a valuable and easy-to-use tool to analyze migration characteristics of meandering channels. Following this introduction, in Section 2, morphological characteristics of the selected study reach of the Madhumati River (Bangladesh) is presented. Bank full geometry and flow properties are estimated from statistical analysis of hydrologic data. Characteristics of bed and bank material are also discussed. In Sec tion 3, linearized 1-D models widely used to simulate meander migration are reviewed and their validities are investigated. Performances of different models are shown in the results and discussion section (Sec. 4). Finally, a summary of conclusions drawn from the present work is presented in Section 5.

2 STUDY AREA

In this section, morphological characteristics of the selected study reach of the Madhumati River is described. Bankfull geometry, flow properties and hydraulic characteristics of bed and bank material are the essential parameters which are responsible for channel changes. Bankfull geometry and flow properties are estimated from statistical analysis of hydrologic data. Hydraulic characteristics of bed and bank material are obtained from the limited available information.

The Madhumati River originates from the Ganges

River and its channel length is about 199 km. It passes through the south-western region in Bangladesh. The location of the study area is shown in Figure 1. It is approximately 1.88 km long. The Madhumati River flowing from the North turns to the left at Mallikpur bend and then flows to the East. The flow directly hits towards the right bank there, and erodes the right bank. Such bank erosion is continuing for many years. Correspondingly, the planform changes year by year. The average meandering bend migration rate in the study area is approximately 35 m/year during the period from 2002 to 2007.

Although there are no gauging stations in the study reach, hydrologic data such as flow discharge could be estimated based on data obtained from the Kamarkhali Transit gauge station which is approximately 30 km upstream of the study reach. This gauging station has been in operation since 2000. Since there are no substantial tributaries between gauge station and the study reach, the flow discharge can be the same as the discharge at the gauge station. Flood hydrographs measured there for different years reveal that they vary from month to month and from year to year. The most important characteristic of the hydrograph is the existence of a long term peak between July and September.

Since floods are the annual events for the river, the dis charge corresponding to 100% exceedance probability can be selected as the bankfull discharge. Therefore, the bankfull discharge is estimated as 2525 m 3 /s. Discharge versus stage relationship is plotted on suitable arithmetic scales from long term monitoring of daily averaged water levels and corresponding dis charges. The array of points almost lay on a curve which is approximately parabolic. A best fit curve is drawn through the points so that the equation can be transformed into the relation between water stage Figure 1. Location of the study area (Mallikpur Bend). Flow is from the North. and spatial average velocity using a definition equation ($\bar{U} = Q/A$). Although there is some scatter, the plot shows that mean velocity increases with the stage. The regression equation allowed for an estimate of the reach averaged mean velocity corresponding to the bankfull stage. The reach averaged mean velocity is found as 1.10 m/s. In the study area, Bangladesh Water Development Board (BWDB) measured cross-sectional shapes at 83 locations in 2006. Cross-sections are spaced approximately 15 m apart. These data are useful for estimating channel geometry.According to averaging procedures, we obtained the followings; 8.00 m for the average water depth, 12.03 m for the average scour depth, 39.93 m for the average distance of deepest point from the right bank, and 26.64 degree for the bank slope at the outer banks. The yearly sediment supply from the banks is not measured directly, but it is estimated that the annual average sediment transport capacity is about 50 million tons in which about 40% are fine sand and the rest consists of silt and clay (Sarker et al., 1999). Fine sands and very fine sands dominate in bottom sediment, but some silt is also present. The median diameter of sediment is 0.179 mm. Although the porosity of bed and bank material is not specified, it could be estimated as 0.44, referring to descriptions in Maidment (1992). 3 APPLICATION OF THE METHODS In 1-D modeling, a channel is represented by a series of nearly equally spaced points representing its centerline and the banks are located at a distance equal to the channel half-width, b on the both sides of the centerline. As the channel width remains same during migration, the left bank, right bank and the channel centerline all move in the same fashion (Johannesson & Parker, 1985). The present paper has thus used the digitized coordinates of the channel centerline in 2002, which is represented by a series of points with Carte sian coordinates.The centerline of the river in the study reach was digitized from satellite images with the aid of Geographic Information System (GIS) ESSRI Soft ware.The coordinates are made dimensionless with the channel half-width, b.

Different linear meander migration models have been proposed to simulate morphodynamic processes of natural meandering rivers. Ikeda et al. (1981) pio neered the theoretical study and numerical simulation of meander migration processes.They developed a lin ear solution to predict near-bank excess velocity based on shallow water flow model. The solution is:

where u is the near bank excess streamwise velocity; χ is the cube root of the ratio of channel length aligned along the valley axis to meandering channel length; C (s) is the dimensionless curvature of the channel cen terline at point s; $C_f = gH_0 I_0 / U_0^2$, in which g is the gravitational acceleration and I_0, H_0, U_0 are the reach averaged values of bed slope, water depth, and flow velocity of the channel aligned along the valley axis;A is the transverse bed slope parameter; F is the Froude number of the channel aligned along the valley axis and s is the dimensionless streamwise coordinate of the channel centerline.

The linear solution for the flow field model is used as an input data for the bank erosion model. Bank erosion rate is assumed proportional to the near-bank excess velocity. A dimensionless erosion coefficient (E_0) is defined as the proportionality constant between the bank erosion rate (ζ) and the near-bank excess velocity (u). Map of time sequential bank line dia grams is used to estimate the erosion coefficient. This model has been validated for rivers in Japan by Hasegawa (1989), in the USA by Johannesson & Parker (1985) and in The Netherlands by Crosato (1989).

Hasegawa (1989) developed a bank migration model based on the integration of the equation of sed iment continuity at the outer bank region in the mean dering channel. The model is based on the assumption that the soil particles on the bank are eroded by the force of flow. The meandering channel migration process is shown in Figure 2.

Hasegawa (1989) provides an analysis of the fac tors controlling bank erosion. Six terms related to transport rate, sediment characteristics, flow proper ties and bank geometry emerge from his analysis. Relative order of magnitude estimation shows that only two terms have first order importance (detailed in Hasegawa, 1989). Among these two terms, one is related to near-bank excess streamwise velocity and thus positively associated with bank erosion.The other term is related to local scour depth. Hasegawa (1989) highlights that local scour depth increases bank height,

Figure 2. Bank migration process: (a) planform of a meandering channel (Johannesson & Parker, 1985); (b) secondary flow is generated due to channel planform curvature; (c) secondary flow carries the near bottom sediments from the outer bank to the inner bank region in the downstream and progressively local scour takes place; (d) the outer bank is undermined and becomes unstable; and (e) unstable bank collapses, and causes bank erosion.

Finally the outer bank migrates. which in turn supply more material to be transported, and in the end slows down erosion. where ζ = migration speed of bank erosion; E_0 = bank erosion parameter; K = constant of bed load transport rate; $* = \tau_{*0} /(\tau_{*0} - \tau_{*c})$, in which τ_{*0}, τ_{*c} = crosssectional average and critical Shields stress; $T = \sqrt{(\tau_{*c} /(\mu_s \mu_k \tau_{*0}))}$, in which μ_s, μ_k = static and dynamic Coulomb friction factor of sediment particles; λ= porosity of bed and bank material; ρ, ρ_s = density of water and sediment particles and η= downward displacement of the bed relative to a cross-sectional mean bed elevation. Three different methods namely methods 1 to 3 are used to predict the location of the channel in 2007. Method 1 is based on Ikeda et al. (1981). Method 2 & 3 follow Hasegawa's model. Method 2 takes into account the 1st term in the right hand side of Equation 2 only, whereas full equation has been used in Method 3. The computed channel centerlines are compared with observed location in 2007. In Method 1, the erosion coefficient is calculated for the period 2002 to 2004 from the map of time sequential bank line diagrams and found equal to 8.10×10^{-7}.

Bankfull geometry and flow parameters, listed in Table 1, are used as input data. The time step equal to 0.25 years is employed in the computations. Therefore, the 5-year period (2002 to 2007) is discretized into 20 time steps. Besides bankfull geometry and flow parameters, Method 2 requires sediment characteristics and transport parameters as listed in Table 2. The erosion coefficient E_0 is estimated equal to 6.65×10^{-6} . The time step is specified as the duration of bankfull discharge of different flood events as explained by Table 1. Bankfull geometry and flow parameters. Parameter Meander Channel Equivalent Straight Channel Bankfull discharge $Q = 2525.00$ m^3/s $Q = 2525.00$ m^3/s Average channel top-width $B =$ 470.00 m $B = 470.00$ m Average channel bed slope $I = 0.00004$ $I_0 = 0.000069$ Average channel depth $H = 8.00$ m $H_0 = 6.655$ m Average flow velocity $U = 1.10$ m/s $U_0 = 1.3223$ m/s Transverse bed slope parameter $A = 6.00$ $A = 6.00$ Froude number $F = 0.12417$ $F = 0.16365$ Friction factor $C_f =$ 0.002594 $C_f = 0.002576$ Table 2. Sediment characteristics and transport parameters. Parameter Value Median diameter of sediment particles, d 0.179 mm Porosity of bed and bank material, λ 0.44 Average transverse slope angle of eroding bank, k 26.64 0 Critical Non-dimensional Shields stress, τ_{*c} 0.073 Static Coulomb friction Coefficient of sediment particles, μ_s 1.00 Dynamic Coulomb friction Coefficient of sediment particles, μ_k 0.70 Constant of bed load transport rate, K 8.00 Density of water, ρ 1000 kg/m^3 Density of sediment particles, ρ_s 2650 kg/m^3

Hasegawa (1989): It is clear that bank erosion does not

occur continuously, but rather during, or in interme

diate response to, high flows. The periods of bankfull

discharge in different flood events are estimated and

five different time steps are taken into computation. In

Method 3, observed near-bank excess flow depths are

used additionally.

4 RESULTS AND DISCUSSION

The author investigated whether meander migration

models are able to reproduce the observed behavior

of the Madhumati River. Three different numerical simulations are carried out using meander migration models; methods 1 to 3.

The results shown in Figure 3 suggest that Method 1 & 2 are able to simulate well lateral migration, how ever these cannot evaluate downstream migration. In these methods bank migration rate is proportional to near-bank excess velocity and the bank erodibility is assumed spatially uniform. In this way both methods have the same erosion law. Near-bank excess velocity depends on channel curvature only. Figure 4 shows that near-bank excess velocities are small in the upstream of bend apex, reach to the maximum value at the bend apex and then decrease in the downstream. Corre spondingly, the bank migration follows the same trend, as shown in Figure 3. In the upstream of bend apex the agreement between predicted and observed migration is good which indicates that the erosion coefficient is reasonably estimated. In the downstream of bend apex near-bank excess velocities are small and the migration thus predicted is underestimated. Such poor prediction can be attributed to the low erosion coefficient. Because other parameters related to bankfull geometry, flow properties and sediment characteristics were assumed uniform.This explains the necessity of imposing higher values of the erosion coefficient for the downstream of bend apex. It would have been easy to use locally different values for the erosion coefficient to obtain perfect agreement, but such a calibration would be misleading. Method 3 is able to simulate the downstream migration well whereas the lateral migration is overestimated (Fig. 3). In this method an influence of the near-bank excess flow depth on the bank migration is added to the Method 2. It is obvious that near-bank excess flow depth influences largely the bank shifting. In the study area a bar is formed inside the bend nearly on the meander axis and water depths are larger near the outer bank. Maximum velocity is predicted at the bend apex whereas the largest water depth is observed downstream of the bend apex. Although the near-bank excess velocities and observed near-bank excess flow depths were out of phase (Fig. 4), they together amplified the bank migration rate. This might be a possible reason for shifting the migration towards the downstream direction. However, this method cannot predict the lateral migration in the upstream of bend apex. It is meaningful to identify the causes of the deviations in order to improve the predictions. It is suggested there are two reasons for the deviations. One possible cause for the deviation was

Figure 3. Simulation of meander migration – Madhumati River (Bangladesh): The digitized coordinates of the channel centerline in 2002 is used as the initial location for simulation. The distance between successive nodes is equal to one channel width. The nodes are numbered to facilitate further discussion in Figure 4. Channel centerlines as predicted in 2007 by method 1, 2 & 3 are shown with the sequence dashed (green), long-dashed (yellow) and long-dash-dotted (blue) lines. The predicted channel centerlines are compared with the observed location in 2007.

considered to be the length scale of spatial discretization. The distance between successive digitized nodes was equal to one channel width (Fig. 3). The curvature at a node was computed with the coordinates of its own, the preceding and the following nodes using a central differentiation scheme.As a result curvature variations between successive nodes are large (Fig. 4). Such large curvature variations increase near-bank excess veloc

ity because the near-bank excess velocity is a function

of local and upstream planform curvatures. Therefore

a simulation was conducted with spatial discretization

of channel half-width equivalent as shown in Figure 5.

The simulation results suggest that the smaller length

scale of spatial discretization does not yield much

better results.

The second and most important reason was consid

ered to be the large flow width of the channel. The

flow width was determined including the very shallow

water zone near the inner bank in the study reach. If

the flow width is estimated to be large in comparison

with the real one, the near bank excess flow depth will

be large. Figure 6 shows that near-bank excess flow

depth effect is much larger than the near-bank excess

velocity on the bank migration rate. As the near-bank
Figure 4. Parameter characteristics: X-axis is position in bend measured downstream in width-equivalent units.Y-axis is non-dimensional local curvature, near-bank excess velocity and near-bank excess flow depth. Non-dimensional near-bank excess velocity, u is predicted by Equation 1 using parameters from Table 1. Local channel curvature is half-width normalized. The near-bank excess flow depth, η is measured and normalized by the average water depth of equivalent straight channel, H_0 . Curvature is defined as positive for anticlockwise turning of river channel, viewing from upstream. Curvature and near-bank excess velocities have opposite sign, therefore bank migration is directed towards the outer bank. Note that node number 25 is the bend apex. Figure 5. Simulation of meander migration – Madhumati River (Bangladesh): The digitized coordinates of the channel centerline in 2002 is used as the initial

location for simulation. The distance between successive nodes is equal to half channel width. Channel centerline as predicted in 2007 by method 3 is shown with the long-dash-dotted (blue) line. The predicted channel centerline is compared with the observed location in 2007. excess flow depth effect was large in the upstream of bend apex, the lateral migration was over predicted. Correspondingly, the prediction of bank shifting will be improved by determining the flow width carefully.

Figure 6. Excess velocity and excess flow depth effect on meander migration: X-axis is position in bend mea sured downstream in width-equivalent units. Y-axis is non-dimensional near-bank excess velocity and near-bank excess flow depth effect. Non-dimensional near-bank excess velocity, u is predicted by Equation 1 using parameters from Table 1. Non-dimensional near-bank excess flow depth effect is computed as the 2nd term in the right hand side of Equation 2.

5 CONCLUSIONS

Migration of meandering channels is discussed using existing models in order to obtain information for river management. Special attention is paid for appli cability of the models to the Madhumati River in Bangladesh as well as for testing validations. The meander migration models proposed by Ikeda et al. (1981) and by Hasegawa (1989) are introduced and applied to channel shifting to test their validities for river management.

Ikeda et al.'s model which was proposed origi nally to evaluate dominant wave lengths of meandering channels is able to discuss the channel migration based on a linear relation between the excess velocity and bank erosion rate and in addition, Hasegawa's model, which is based on the same linear relation as Ikeda

Employing surrogate modelling for the calibration of a 2D flood simulation model

V. Christelis, V. Bellos & G. Tsakiris

Laboratory of Reclamation Works and Water Resources Management & Centre for the Assessment of Natural Hazards and Proactive Planning, School of Rural and Surveying Engineering, National Technical University of Athens, Greece

ABSTRACT: In this paper, the automatic calibration of the in-house FLOW-R2D hydrodynamic model is addressed, for a real world flood event created after the Tous dam break (Spain). The Manning coefficient in the entire computational domain and the effective slope of the upstream boundary cross-section (for deriving the upstream boundary steady flow condition) are selected for calibration. Due to the long simulation run times of hydrodynamic models, conventional optimisation techniques cannot be easily implemented for automatic calibration while the commonly applied trial and error method has several limitations. To cope with the excess computational burden, a surrogate-assisted optimisation algorithm (namely, MLMSRBF) is employed in this paper. To set a benchmark solution, a FLOW-R2D-based automatic calibration is also performed and the results are compared to the MLMSRBF method. Results indicate that for a computational budget of 100 FLOW

R2D simulations, both the conventional and the surrogate-assisted algorithm perform equally well while they

outperform the traditional trial and error approach.

1 INTRODUCTION

1.1 Basic notions

The automatic calibration of 2D flood model parame ters in real world applications is of practical interest to hydraulic modelers.Yet, it is a time consuming process due to the large computational cost of each simula tion run. Conventional or evolutionary optimisation techniques cannot be implemented in reasonable com putational times, even if each single simulation run is reduced to the order of some minutes. Therefore, up to now the dominant approach is the trial and error method which though has several disadvantages. Among these are the subjective decisions by the model user, and the limited number of simulation runs which can be performed, whereas the optimal search space cannot be sufficiently explored.

One of the available methods to cope with the excess computational burden, arising from 2D flood model simulations, is to employ surrogate modelling tech niques. In this paper, a surrogate modelling method, known as Multistart Local Metric Stochastic Radial Basis Function (MLMSRBF), is utilised for the auto matic calibration of the two parameters required in the 2D flood simulation model called FLOW-R2D.

The case study used in this paper for illustrating the proposed methodology, refers to the flood event created after the Tous dam break in Spain. The first parameter which is calibrated is the Manning friction coefficient, needed in the friction model, for the entire computational domain. Additionally, the effective slope of the upstream boundary cross-section required for deriving the upstream boundary steady flow condition, is also calibrated.These two factors are among others (topographic measurement errors, building representation in urban environments, grid size etc) which create uncertainties to the numerical results derived by the implementation of a model (Abily et al. 2016). The calibration results from the MLMSRBF method are compared to the results derived by a FLOW-R2D-based calibration using a well-tested evolutionary algorithm within a limited computational budget. 1.2 FLOW-R2D model The FLOW-R2D model is an in-house numerical model, which solves the full dynamic form of the 2D Shallow Water Equations (2D-SWE) through the Finite Difference Method and the explicit McCormack numerical scheme (McCormack 1969), in a cell-centered, non-staggered computational grid. The wet/dry bed modelling is achieved through a water depth threshold (h_{dry}) which distinguishes wet and dry cells. Besides, artificial viscosity is added in the context of the 2D-SWE discretisation, through a diffusion factor (ω). From the existing options of friction modelling, the empirical Manning equation was adopted. A detailed presentation of the model can be found at Tsakiris & Bellos (2014) whereas interesting applications of the model are also available at Bellos & Tsakiris 2015a, b.

1.3 The surrogate modelling

The time-intensive numerical simulations associated with hydrodynamic models, overwhelm the applica tion of simulation-optimisation methods for automatic calibration approaches. Surrogate modelling tech niques may be used in such cases in order to alleviate the overall computational burden and to approximate a good solution within a reasonable computational time. That is, efficient approximate models, com monly called surrogate models or meta-models, are constructed based on a relative small number of runs (initial sampling plan) of the computationally expen sive numerical model in order to predict the responses of the latter for unseen data (Forrester et al. 2009; Razavi et al. 2012).

Despite of the fact that the implementation of sur rogate models significantly reduces the computational cost for each objective function evaluation, false or far from optimal solutions may be introduced by sur rogate models during optimisation tasks (Jin 2011). Therefore, a strategy is typically formulated in which new sample points are added to the initial sampling plan to improve the predictive capabilities of the sur rogate model. The most comprehensive approach is to add infill or update points to the initial sampling plan using an infill strategy which aims at improving both the local and global accuracy of the surrogate model (Forrester et al. 2008).

Here, the Multistart Local Metric Stochastic RBF (MLMSRBF) algorithm, developed by Regis & Shoe maker (2007), is used to accomplish the automatic calibration task of the FLOW-R2D model. The MLM SRBF algorithm utilises a cubic RBF surrogate model with a linear polynomial tail, in order to approximate a computationally "expensive" black-box function (here, the FLOW-R2D numerical model). The MLM SRBF algorithm applies a symmetric Latin Hypercube sampling design to create the initial sampling plan for constructing the RBF surrogate model. The update points for evaluation with the original numerical model are selected based on the approximated objective func tion value obtained by the RBF model and on a weighted distance criterion related to previously eval uated points. Detailed description of this procedure is presented elsewhere (Regis & Shoemaker 2007).

It should be noted that RBF surrogate models have gained wide acceptance in the field of engineer ing optimisation during the last decade (e.g. Regis & Shoemaker 2004; Wild et al. 2008; Jakobsson et al. 2009; Regis 2011; Rashid et al. 2013; Regis 2013; Regis 2014). An advantageous characteristic of RBF surrogate models is that they can deal with nonlinear responses, a common case in engineering optimisation, while they still preserve a rather sim

ple formulation (Forrester et al. 2008; Sóbester et al.

2014). Recently, RBF surrogate models have been

successfully applied in the field of water resources management (e.g. Mugunthan et al. 2005; Shoemaker et al 2007; Razavi et al., 2012; Christelis & Mantoglou 2015; Tsoukalas et al. 2016). 2 CALIBRATION 2.1 Real world case study An extreme storm episode, which occurred during the 20th and 21st of October 1982 (500 mm rain in 24 h), hitValencia, Spain.The rock-fillTous dam, built on the Júcar river, failed at 19:15 on the 20th of October with destructive consequences in the downstream area and especially in the Sumacárcel town located just 5 km downstream of the dam (Figure 1). Details of the catastrophic flood event were collected in the framework of the IMPACT research project (Alcrudo & Mulet 2007). Among these data, the maximum inundation depths recorded at 21 gauges in Sumacárcel town are available. It should be mentioned that various researchers have tested their algorithms using the data from this extreme event (Abderrezzak et al. 2008; Bellos & Tsakiris 2015c). 2.2 Numerical simulation As mentioned above, the two parameters which should be calibrated for the numerical simulation are the Manning friction coefficient of the entire computational field and the effective slope S eff which is incorporated at the upstream boundaries (steady flow condition). Despite of the fact that two different friction zones are proposed by Alcrudo & Mulet (2007), (one with orange trees and one for the rest of the computational area), in this study the entire computational field is assumed to be in a unique friction zone. This is chosen because all the gauges which recorded the flood event are located in the second friction zone. For computational reasons, only a part of the entire computational domain is analysed for the simulation of this event (Figure 2). The selection of the space for analysis is made taking into account that the main interest is focused on the town and especially near the gauges in which data were recorded. This made the comparison viable between numerical and historical data. In this study the size of each computational cell is 5 × 5 m and the entire computational area is composed of 36337 cells. The wet/dry threshold (h dry = 0.01 m), the Courant number (CFL = 0.05) and the diffusion factor (ω= 0.9) are assumed constant in order to reduce the computational burden of the calibration phase. The Courant number and the diffusion factor are selected based on the authors' experience aiming at reducing oscillations and avoiding non-realistic errors in the results.This is achieved by a preliminary trial and error method at the initial steps of the study. For the upstream boundary conditions, steady state and specifically the peak flow equal to 15000 m^3/s is chosen rather than the entire flood hydrograph. This

Figure 1. Location of the case study area (Sumacárcel town).

is because the computational cost in the latter case becomes prohibitive even for the calibration phase. The water depth in the gauges can be considered as stable, after 700 seconds simulation.

For the building representation, the solid boundary condition is used through a modified reflection boundary method (Bellos & Tsakiris 2015a), which seems to produce better results, with lower computational cost. The lateral boundaries of the computational domain are considered also as solid boundaries, for the mass balance conservation. For the downstream boundaries the open boundary condition is used. No base flow is considered as initial condition.

A detailed presentation of the numerical simulation with calibration by the trial and error method can be found in a previous work (Bellos & Tsakiris 2015c).

2.3 Results

The automatic calibration of FLOW-R2D belongs to the box-constrained optimisation problem of the following form:

in which f represents the objective function to be min

imised which in our work is the sum of the Square Error
Figure 2. Computational domain and location of the gauges.
(SE S) between the numerical results and the historical data of the 21 gauges and θ is the vector of FLOWR2D model parameters [n,S eff] which are bounded in the [a,b]. The lower and upper limits for the vectors a, b are pre-selected for each model parameter while for a better performance of the surrogate optimisation algorithm MLMSRBF, the parameter space is mapped to [0,1] (Regis & Shoemaker 2007). The determination of the vector [a,b] for each calibrated parameter is based on the authors' experience. The vector values cover a realistic range whereas a significant part of the space is searched, for both of the above methodologies. Specifically, the range [0.0001,0.02] for the effective slope S eff , and [0.02,0.25] for the Manning coefficient n, are selected. A maximum number of 100 objective function evaluations with the FLOW-R2D model were allowed. It should be noted that each simulation lasts for about 1.5 h using an i7 CPU/3.6 GHz PC. By using the same computational budget of 100 FLOW-R2D runs, the above optimisation problem is also solved using the FLOW-R2D model alone. For this second optimisation approach the Evolutionary Annealing-Simplex (EAS) algorithm has been applied (Efstratiadis & Koutsoyiannis 2002). EAS combines the concepts of evolutionary search, downhill simplex scheme and simulated annealing and it has been successfully applied in several problems of automatic calibration of hydrological models (Rozos et al. 2004; Efstratiadis et al. 2014) as well as in coastal aquifer management (Christelis & Mantoglou 2016). Detailed principles and steps of EAS algorithm can be found in Efstratiadis & Koutsoyiannis (2002) and Rozos et al. (2004). Note that the initial sampling plan produced by the MLMSRBF algorithm is also used as an initial population for the EAS algorithm for achieving a fair comparison between them, since the initial conditions affect the stochastic nature of both algorithms.

Table 1. Calibrated parameters for the best value of the

objective function after 100 simulations. Effective Manning's Square Slope Coefficient Error

Methodology	S eff	n	SE s
MLMSRBF	0.019	0.194	37.889

EAS 0.020 0.201 37.876

Figure 3. Evolution of the SE S values for the MLMSRBF and the EAS algorithm.

Additionally, five independent optimisation runs are performed for each approach. A comparison between the two optimisation approaches is presented below. In the Table 1, the SE S values and the values of the calibrated parameters derived by the above procedure are presented. In Figure 3, the evolution of the SE S values against the number of simulations is plotted for both the MLMSRBF and the EAS algorithm. In Fig ures 4 to 6, a comparison is made between the observed data and the numerical results derived from the FLOW R2D model using the calibrated parameters obtained by the MLMSRBF and the EAS algorithm. It should be noted that in the gauges where the observed historical maximum water depth exhibits a range of values, an average value is used (Alcrudo & Mulet 2007). Finally, in Figures 7 and 8, the flood hazard maps derived by the FLOW-R2D model are presented. FLOW-R2D uses the calibrated parameters obtained by the MLMSRBF algorithm.

3 CONCLUDING REMARKS

3.1 Discussion

Based on the results derived from the calibration approach, it seems that both MLMSRBF and EAS

algorithms perform equally well for a computational

budget of 100 FLOW-R2D simulations.

The automatic calibration approach significantly

improves the simulation results. In a previous work,

the best SE S value, derived from the trial and error
Figure 4. Numerical results compared to observed data for the MLMSRBF and the EAS methods (Gauges 1–7). Figure 5. Numerical results compared to observed data for the MLMSRBF and the EAS methods (Gauges 8–14). Figure 6. Numerical results compared to observed data for the MLMSRBF and the EAS methods (Gauges 15–21). method, was 47.56, while with the automatic calibration approach the error is decreased by about 20% (Bellos & Tsakiris 2015c). In terms of the derived parameter values, the Manning's coefficient takes significantly high values according to the related literature. For example, this coefficient ranges from 0.03 s/m 1/3 for natural streams, to 0.15 s/m 1/3 for floodplains with trees (Chaudhry 2008; Chanson 2004). However, various researchers claim that the Manning's coefficient is in fact greater than that proposed in real world conditions, since they incorporate turbulence energy losses in the

Figure 7. Flood hazard map (water depths) derived with the

FLOW-R2D model and the calibrated parameters obtained by

the MLMSRBF algorithm.

Figure 8. Flood hazard map (flow velocities) derived

with the FLOW-R2D model and the calibrated parameters

obtained by the MLMSRBF algorithm.

micro-scale (Jarrett 1985). According to our results,

probably these coefficients should be determined more

carefully from the modellers. Furthermore, the effec

tive slope is in fact the energy slope of the flow at the upstream boundaries and thus it is a black box param eter. In view of this, the calibrated values cannot be estimated whether are realistic or not.

A series of preliminary simulation-optimisation runs was performed by using two friction zones for the site under study, as suggested by Alcrudo & Mulet (2007). However, the resulting manning coefficient for the ungauged orange trees field was also unreal istic, compared to the expected literature values, thus rendered the parameterization in two distinct friction zones unnecessary. Note that due to the absence of water level measurements in the orange trees field the objective function values are mainly driven by the error between the simulated values and the gauged val ues from the rest of the computational field. Based on the above, we opt for a single manning coefficient for the entire computational field in order to reduce the

Jakobsson, S., Patriksson, M., Rudholm, J., Wojciechowski, A. 2010. A method for simulation based optimization using radial basis functions. Optimization and Engineer ing 11(4): 501–532.

Jarrett, R. 1985. Determination of roughness coefficients for streams in Colorado. Water Resources Investigations Report 85-4004, US Geological Survey.

Jin, Y. 2011. Surrogate-assisted evolutionary computation:

Recent advances and future challenges. Swarm and Evo

lutionary Computation 1:61–70.

Chanson, H. 2004. The hydraulics of open channel flow: an

introduction, 2nd edition. Elsevier editions.

Chaudhry, M.H. 2008. Open channel flow, 2nd edition.

Springer editions.

Forrester A.I.J., Sôbester, A., Keane, A.J. 2008.
Engineering

design via surrogate modelling-A practical guide. Wiley,

editions.

Forrester A.I.J., Keane, A.J. 2009. Recent advances in

surrogate-based optimization. Progress in Aerospace Sci

ences 49: 50–79.

McCormack, R.W. 1969. The effect of viscosity in hyper

velocity impact cratering. AIAA hypervelocity impact

conference, Cincinnati: Paper 69–354.

Mugunthan, P., Shoemaker, C.A., Regis, R.G. 2005.

Comparison of function approximation, heuristic and

derivative-based methods for automatic calibration of

computationally expensive groundwater bioremediation

models. Water Resources Research 41: W11427, doi:

10.1029/2005WR004134.

Rashid, K.,Ambani, S., Cetinkaya, E. 2013.An adaptive mul

tiquadric radial basis function method for expensive black

box mixed-integer nonlinear constrained optimization.

Engineering Optimization 45(2): 185–206.

Razavi, S., Tolson, B.A., Burn, D.H. 2012. Numerical

assessment of metamodelling strategies in computation

ally intensive optimization. Environmental Modelling &

Software 34: 67–86.

Razavi, S., Tolson, B.A., Burn, D.H. 2012. Review of sur

rogate modeling in water resources. Water Resources

Research 48(7): W07401, doi: 10.1029/2011WR011527.

Regis, R.G. & Shoemaker, C.A. 2004. Local function approx

imation in evolutionary algorithms for the optimization

of costly functions. IEEE Transactions on Evolutionary

Computation 8(5): 490–505. Regis, R.G. & Shoemaker, C.A. 2007. A stochastic radial basis function method for the global optimization of expensive functions. INFORMS Journal on Computing 19(4): 497–509. Regis, R.G. 2011. Stochastic radial basis function algorithms for large-scale optimization involving expensive black-box objective and constraint functions. Computers & Operations Research 38: 837–853. Regis, R.G. & Shoemaker, C.A. 2013. Combining radial basis function surrogates and dynamic coordinate search in high-dimensional expensive black-box optimization. Engineering Optimization 45(5): 529–555. Regis, R.G. 2014. Evolutionary programming for highdimensional constrained expensive black-box optimization using radial basis functions. IEEE Transactions on Evolutionary Computation 18(3): 326–347. Rozos, E., Efstratiadis, A., Nalbantis, I., Koutsoyiannis, D. 2004. Calibration of a semi-distributed model for conjunctive simulation of surface and groundwater flows. Hydrological Science Journal 49(5): 819–842. Shoemaker, C.A., Regis, R.G. Fleming, R.C. 2007. Watershed calibration using multistart local optimization and evolutionary optimization with radial basis function approximation. Hydrological Science Journal 52(3): 450– 465. Sôbester, A., Forrester, A.I.J., Toal, D.J.J., Tresidder, E., Tucker, S. 2014. Engineering design applications of surrogate-assisted optimization techniques. Engineering Optimization 15(1): 243–265. Tsakiris, G. & Bellos, V. 2014. A numerical model for twodimensional flood routing in complex terrains. Water Resources Management 28(5): 1277–1291. Tsoukalas, I., Kossieris, P., Efstratiadis, A., Makropoulos, C. 2016. Surrogate-enhanced evolutionary annealing simplex algorithm for effective and efficient optimization of water resources problems on a budget. Environmental Modelling & Software, 77: 122–142. Wild, S.M., Regis, R.G., Shoemaker, C.A. 2008. ORBIT: Optimization by radial basis function interpolation in trust-regions. SIAM Journal on Scientific Computing 30(6): 3197–3219. Zech, Y. & Soares-Frazao, S. 2007. Dam-break flow experiments and real-case data. A database from the European IMPACT research. Journal of Hydraulic Research 45: 5–7. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Estimating stem-scale mixing coefficients in low velocity flows

F. Sonnenwald

Department of Civil and Structural Engineering, University of Sheffield, Sheffield, UK

I. Guymer, A. Marchant & N. Wilson

School of Engineering, University of Warwick, Coventry, UK

M. Golzar & V. Stovin

Department of Civil and Structural Engineering, University of Sheffield, Sheffield, UK

ABSTRACT: Stormwater ponds are SuDS devices intended to moderate the negative environmental impacts of

stormwater run-off. A current, joint, research programme is investigating the effects of heterogeneous vegetation

distributions in stormwater ponds and developing Computational Fluid Dynamics (CFD) techniques to simulate

3D solute transport processes in low velocity flows. The aim of the project is to generate a unique dataset that

describes the influence of different types and configurations of vegetation on the pond’s fundamental flow – and

treatment – characteristics. This characterisation can then be used to evaluate existing run-off treatment ponds

that may be delivering sub-optimal levels of treatment.This paper presents results from an initial laboratory study,

with regular uniform emergent artificial vegetation, from which longitudinal dispersion coefficients have been

obtained over a range of target flow velocities in a narrow (Armfield) flume. These have been integrated with

stem-scale CFD derived estimates of transverse variation of mean longitudinal velocity across a vegetation cell to

investigate the ability of a Chikwendu (1986) n-zone type approach to predict transverse dispersion coefficients

over a scale suitable for inclusion in future 3D CFD pond models. Assuming that the small stem-scale variations

in velocity form a repeating pattern at the patch scale, and that stem-scale transverse dispersion effects integrate,

this approach has been successfully applied to predict transverse and longitudinal mixing in a wide flume solely

with longitudinal parameters estimated from a narrow flume.

1 INTRODUCTION

Stormwater run-off typically contains and transports

a wide range of pollutants, resulting in negative envi

ronmental effects with potential threats to ecosystems

and health. In the UK, hundreds of run-off treat

ment (stormwater) ponds intended to moderate these

impacts are likely to be delivering sub-optimal levels

of improvement in water quality due to poor under

standing of flow patterns and the effects of vegetation.

To further develop this understanding a current, joint,

research programme is investigating the effects of het

erogeneous vegetation distributions on low velocity flows, conducting laboratory work has been carried out to quantify mixing around uniform emergent artificial vegetation. This paper aims to show that, for future inclusion into 3D Computational Fluid Dynamics (CFD) pond models, transverse dispersion may be esti mated from longitudinal dispersion and a transverse profile of longitudinal velocity using a Chikwendu (1986) n-zone type approach.An expression to achieve this in an infinite patch of vegetation is derived, and experimental and computational results are used to demonstrate its applicability. 1.1 Mixing Here, ‘mixing’ refers to the combination of the processes of diffusion, advection, and dispersion by which neutrally buoyant particles in a flow field are transported. Many engineers will be familiar with the onedimensional Advection-Dispersion Equation (ADE), where C is the cross-sectional average concentration (e.g. of a dye or tracer), t is time, u is the mean longitudinal velocity, x is the position along a channel, and D x is the longitudinal mixing coefficient that describes the spread of the tracer (Rutherford 1994). D x , interestingly, is an additive term that combines the effects of several different processes at different scales. In uniform emergent vegetated flows these processes include: molecular diffusion at the smallest scale (generally ignored); stem-scale wake turbulence; and at the largest scale the effects of differential advection, where flow speeds up as it passes around stems (Nepf 1999, Nepf 2012).

Ponds can have a variety of geometries and are often non-linear in shape, making them complex three-dimensional systems, often simulated as two dimensional models (Persson 2000). To simulate mix ing processes within stormwater ponds, the complex geometries require that dispersion around vegetation be characterised at least longitudinally and trans versely. In this 2D scenario, turbulence effects act to promote mixing in both directions, while the differen tial advection effects act primarily in the direction of flow. Dispersion is not the same in both directions. Despite the need for at least 2D mixing characteri sation in ponds, most laboratory systems are designed to collect cross-sectionally well-mixed dispersion data suitable for application to a 1D model only. Chikwendu (1986) presents a model relating transverse and longi tudinal dispersion through the transverse variation of longitudinal velocity,

where q_j is the fractional thickness of layer (layer thick ness h_j over total thickness h), $u_{12...j}$ is the mean veloc ity of layers 1 to j, D_{xj} is the longitudinal dispersion within layer j, and

with $D_{yj(j+1)}$ being the transverse dispersion between layers j and $j + 1$. This suggests that a characterisa tion of longitudinal dispersion and transverse variation of longitudinal velocity from a laboratory system designed to collect cross-sectionally well-mixed lon gitudinal dispersion data could be used to approximate transverse dispersion coefficients.

1.2 Computational fluid dynamics

In a CFD model, the Navier-Stokes equations (describ

ing forces acting on a volume of water) are solved

iteratively to resolve flow-field characteristics for any

arbitrary geometry (Versteeg & Malalasekera 2007).

In addition to the mass and momentum equations,

there is a turbulence modelling component to most

CFD simulations that is necessary to estimate the

shear stresses due to turbulent dissipation. Turbu

lence modelling is generally based on the concept of

'Reynolds Averaging', where the CFD model does not

calculate the turbulent fluctuations and instead uses

approximations to estimate their effect on the mean

flow characteristics. This gives the Reynolds Averaged

Navier-Stokes, or RANS, equations.

Nominally, dispersion could be estimated directly

from a CFD model from the turbulent viscosity parameter over the Schmidt number ($D = \mu_t / S_{ct}$). The former describes the momentum transport due to turbulence and the latter the ratio of momentum to mass transport. However, this approach cannot be used due to uncertainties and limitations of most turbulence models, e.g. isotropic μ_t implying isotropic solute dispersion. Nonetheless, this paper aims to demonstrate the usefulness of stem-scale CFD modelling for obtaining velocity data that may then be utilised to estimate mixing in vegetation in combination with laboratory experiments. 2 METHODOLOGY 2.1 Laboratory data collection A 15 m long by 30 cm wide recirculating narrow Armfield flume, a laboratory system designed to collect cross-sectionally averaged longitudinal dispersion data, was fitted with uniform emergent artificial vegetation at the University of Warwick (Fig. 1). The vegetation was made of 4 mm diameter drinking straws fitted to a 65 × 30 cm baseplate in a staggered grid, with stems being separated by 5 cm transversely and 2.5 cm longitudinally for a density of 333 stems/m 2 (frontal facing area a = 1.592 m −1 , solid volume fraction φ = 0.005). Every other row of straws is offset transversely by 5 cm to form a hexagonal grid. A single repeating ‘cell’ of the vegetation can be defined to be a 5 cm by 10 cm rectangle with a quarter of a straw in each corner and a full straw in the centre (Fig. 2). Flow enters and leaves along the 10 cm edge. Multiple baseplates were placed along the entire length of the flume. The baseplates were designed so that the centre of the channel coincides with a line of stem. Due to the width of the flume, this would mean that a stem should coincide with the channel wall, but no half-stems were placed at the channel wall, instead assuming that the friction generated by the wall would be similar to that generated by the stems in an infinitely wide patch of vegetation. Figure 1. Artificial vegetation located in the Armfield flume at the University of Warwick.

Figure 2. A schematic of an artificial vegetation baseplate

(plan-view), showing dimensions, example Cyclops fluorom

eter positioning, and the repeating vegetation ‘cell’. Flow is

left to right.

Figure 3. Data record from fluorometers after

pre-processing, target velocity of 17 mms −1 . Sub-sampled

for presentation.

Table 1. Target velocity and optimised mean longitudinal

velocity.

Experiment	Target velocity (mms −1)	Optimised u (mms −1)	Re d
1	7	7.23	29
2	10	10.16	41
3	13	14.64	59
4	17	17.09	68
5	20	20.60	82
6	30	29.37	117

7 40 40.14 161

8 50 51.65 207

Eight flow rates with target velocities of 7, 10, 13, 17, 20, 30, 40, and 50 mms -1 were tested with a 0.015 m flow depth. These velocities cover the range of laminar to turbulent flow (expected to be in ponds) based on Re d = ud/ν, where d is stem diameter. Table 1 shows approximate stem Reynolds numbers, where Re d $>\approx 200$ should be fully turbulent flow (Nepf 1999). The slope of the flume was adjusted in conjunction with its tailgate to establish uniform flow. A diffuser plate was placed just upstream of the vegetation to help straighten the flow.

Tracing was carried out by injecting dye directly into the upstream pipe ahead of its outlet into the flume. Dye concentration was recorded at 5 Hz using four Turner Designs Cyclops 7 fluorometers located

centre channel (i.e. replacing stems) and mid-flow depth at 4.1, 6.6, 7.1, and 9.6 m downstream of the diffuser plate. The combination of the four instruments results in 3 reaches, instrument 1 to 2, 2 to 3, and 3 to 4. Three repeat injections were carried out at each target velocity. For each reach and repeat injection, optimised velocity and longitudinal dispersion coefficients were calculated using a similar approach to Guymer & O’Brien (2000). Background concentration levels were subtracted as the mean of the first 5 minutes of the trace. The start and end of the trace were defined as the closest 5 minutes to the location of the peak of the trace where the mean value was below 1% of the peak concentration. Mass-balance was then assumed and the MATLAB (The MathWorks Inc. 2015) lsqcurvefit function used to minimise the sum-of-squares difference between a predicted and the recorded downstream profile, varying u and D_x. The predicted downstream concentration profile was calculated using the standard routing procedure of Rutherford (1994), where $C(x_1, t)$ is the upstream concentration profile, $C(x_2, t)$ is the downstream predicted profile, $\bar{t}$ is the travel time (difference between centroids), and τ is an integration variable. An example of the collected dye data after pre-processing is shown in Figure 3. 2.2 CFD modelling Due to limitations of the available velocity probes, 2D Computational Fluid Dynamics (CFD) modelling has been carried out to estimate the transverse variation of the longitudinal velocity. A single vegetation cell (which represents any arbitrary location within the vegetation or flume) has been modelled in ANSYS Fluent 15 (ANSYS, Inc. 2014) and meshed with a triangular 0.1 mm mesh for a total of 1,041,708 cells (verified to be mesh independent).The ‘inlet’and ‘outlet’ of the cell (i.e. faces perpendicular to the direction of flow) have been modelled as a periodic boundary with a specified mass flow rate. The ‘side walls’ of the cell (parallel to the direction of flow) have been modelled as symmetry boundaries. These conditions represent an idealised infinite patch of vegetation and therefore neglect the effects of the side-walls and the bed. Due to the low velocities and small cells involved, the ‘Enhanced Wall Treatment’ option was specified for near-wall modelling. Second order discretisation was used for all terms for accuracy and the ‘Coupled’ pressure-velocity coupling was used to increase model stability. Models were run with the three most common turbulence models; the $k-\varepsilon$, $k-\omega$, and Reynolds Stress Model (RSM) models. In total, 24 models were run, one for each of the three turbulence models

combined with 8 for each of the experimental tar

get velocities. Mass flow rate for each experimental

target velocity was specified to match the optimised

mean velocity. Transverse profiles of flow-field char

acteristics with points every 0.2 mm in the y-direction

were extracted longitudinally every 0.2 mm in the x

direction of the vegetation cell to form a 0.2 by 0.2 mm

resolution grid.

2.3 Applying Chikwendu (1986)

To calculate transverse dispersion D_y from the available data, (2) may be simplified given the following assumptions: that the layer thickness is uniform; that the approach of Chikwendu (1986) can be applied at the smaller vegetation cell scale and integrated to give the effects at the patch scale; that transverse dispersion may therefore be represented by a single value at the patch scale; and that turbulent dispersion is the primary process governing transverse dispersion. In this scenario, the first two terms in (2), which represent a position dependent relative contribution of the velocity shear to the dispersion, simplify to an average value of 1/30. This then gives

where h is the width of the vegetation cell (here 0.1 m) and q is the fractional velocity layer thickness (here $2 \times 10^{-4}/0.1 = 0.002$). This may be rearranged as

to evaluate the transverse dispersion given longitudinal dispersion and a regularly spaced repeating profile of transverse variation of longitudinal velocity.

2.4 Validation

Linear trends with velocity have been fitted to the longitudinal dispersion coefficients optimised from the recorded Armfield flume data and transverse dispersion coefficients estimated from (6). West et al. (2016) present a laboratory Laser Induced Fluores cence (LIF) wide flume experimental system, created at the University of Warwick, to record solute trace data varying in time and transversely simultaneously. They recorded data 1 m and 2 m from a pulse injection travelling through the same artificial vegetation as previously described, again assumed to represent an infinite patch of emergent vegetation. Using centroid to centroid estimates of u from the West et al. (2016) data, values of D_x and D_y were estimated from the linear trends and used to make wide flume downstream predictions from their recorded upstream profiles.

To make the predictions, a 2D routing procedure (equivalent to (4)) was used. This is described by Baek et al. (2006), where W is the channel width and ψ is an integration variable. West (2016) recorded profiles of transverse variation in longitudinal velocity with Ultrasound Velocimetry Profiling (UVP) probes in the same wide flume experimental system. These have been used to validate the CFD velocity predictions. The quality of the predicted dispersion coefficients has been evaluated using the R^2_t goodness-of-fit correlation metric to compare recorded and predicted solute traces, where 1 indicates a perfect fit and values less than or equal to 0 indicate no fit (Young et al. 1980). 3 RESULTS The recorded trace data from the Armfield flume was high-quality, although slightly noisy due to the high sampling frequency. This is visible in Figure 3. Table 1 shows a comparison of the target configuration velocity compared to the optimised mean longitudinal velocity. Though there is minor variation between the two, the experimental results reflect the intended configurations. 3.1 Longitudinal dispersion The mean R^2_t value from comparing the recorded solute traces from the Armfield flume with optimised downstream predictions was 0.9994 with a standard deviation of 0.0004, indicating all of the optimised profiles are good fits to the recorded data. Figure 4 shows the mean optimised longitudinal dispersion coefficients (across all reaches) plotted against mean optimised longitudinal velocity for each experimental configuration. The optimised dispersion coefficients show a linear trend with velocity, with all data points being near the best-fit linear trend-line. 3.2 CFD modelling To calculate transverse dispersion using (6), it is necessary to first evaluate the transverse variation of longitudinal velocity. Figure 5 shows contours of u for the three different turbulence models, k − ε, k −ω, and RSM, alongside mean profiles (across the length

Figure 4. Mean optimised longitudinal dispersion coeffi cients (o), vertical and horizontal bars (mostly inside the points) indicate 95% confidence intervals. Best fit lin ear trend with $R^2_t = 0.9944$ and $D_x = 2.4534 \times 10^{-2} u + 1.1156 \times 10^{-4}$.

Figure 5. Contour plots of u for a 'vegetation cell' for the three turbulence models with mean profiles of transverse variation in longitudinal velocity (right). White circles show stem positions, crosses (blue) show experimental results by West (2016). Brighter intensities equate to higher velocities, with a peak u of 25.26 mms $^{-1}$. Solid line is k − ε (red), dotted line is k −ω (yellow), and dashed line is RSM (purple). Flow is left to right with a target velocity of u = 17 mms $^{-1}$.

of the vegetation cell) of transverse variation of longi tudinal velocity from CFD and the experimental wide flume results of West (2016). The k −ω and RSM models show evidence of eddy shedding from the cen tral stem. This is not entirely unexpected as the target velocity of 17 mms $^{-1}$ corresponds to $Re_d \approx 68$, which

Nepf (1999) suggests is above the value of $Re_d \approx 50$ where eddy shedding may start to occur. The eddy shedding is not apparent at the corner stems due to the symmetry boundary condition.

Despite the eddy shedding, the variation between turbulence models in the mean transverse profiles of longitudinal velocity over the vegetation cell is very small. This suggests that for the purposes of estimating longitudinal velocities here, the choice of turbulence model is not important. Encouragingly, the CFD-generated mean velocity profiles are also compa rable to the experimentally recorded velocity profile. Some of the difference between the experimental pro file and CFD profiles is likely to be due to the fact that the experimental profile is a single line measurement, while the CFD profile is a mean measurement over the length of the vegetation cell in the direction of flow.

The more rounded shape of the experimental profile Figure 6. Contour plots of μ_t for a 'vegetation cell'showing the relative intensities of μ_t for the three turbulence models with transverse profiles of longitudinal mean μ_t (right). White circles show stem positions. Brighter intensities equate to greater values of μ_t, with a peak value of 0.9213 $cm^2 s^{-1}$. Solid line is $k-\varepsilon$ (blue), dotted line is $k-\omega$ (red), and dashed line is RSM (yellow). Flow is left to right with a target velocity of $u = 17$ mms^{-1}. Figure 7. Estimated transverse dispersion coefficients (o) based on optimised mean velocities. Best fit linear trend with $R^2_t = 0.9733$ and $D_y = 4.1203 \times 10^{-4} u - 4.8434 \times 10^{-6}$. The vertical dotted line occurs at $Re_d = 100$. may also suggest that the models are under-estimating turbulence as higher turbulence levels have increased momentum diffusion that should tend to cause a more uniform velocity profile. The velocity profiles generated by the $k - \varepsilon$ model have been used in calculating transverse dispersion as the lack of eddy shedding results in a more stable numerical solution. Figure 6 shows contours of μ_t and here the differences between the turbulence models are more apparent, with the $k - \omega$ model indicating practically no turbulence and the $k - \varepsilon$ indicating more turbulence behind a stem than the other models. The differences in this case lead to a lack of confidence in using pure CFD modelling for calculating dispersion, as choice of model can heavily impact on the relevant flow characteristics. Estimating dispersion coefficient purely based on CFD results does not appear to be practicable at this time. 3.3 Transverse dispersion Figure 7 shows transverse dispersion coefficients, estimated using (6), plotted against optimised longitudinal velocity. Nepf (1999) suggests that D_y should follow a linear trend with respect to u and so a linear trend line

Figure 8. A sample downstream prediction (contours) created using (7) compared to recorded downstream wide flume data

from West et al. (2016). Estimated $D_x = 5.5090 \times 10^{-4}$ m^2 s^{-1}, estimated $D_y = 1.2222 \times 10^{-5}$ m^2 s^{-1}, and $u = 17.9$ mms^{-1}

calculated centroid to centroid of the wide flume data. $R^2_t = 0.7901$.

has been fitted. However, there appears to be some significant variation about the line and instead it appears that there are two linear trends, one occurring above $u = 0.02$ ms^{-1} ($Re_d = 100$) and one below. This suggests a change in the flow physics governing transverse dispersion. Potentially the transition to fully turbulent flow is occurring at a lower Reynolds number than $Re_d \approx 200$, but more likely the eddies being shed by neighbouring stems are interacting (Tanino & Nepf 2008) in such a way that dispersion is increased similarly to the effects of fully turbulent flow past stems.

Without a more detailed investigation into the relevant flow physics to suggest otherwise, the linear fit has been used for simplicity.

3.4 Wide flume predictions

The experimental wide flume data of West et al. (2016) were recorded through the same vegetation and therefore estimates of dispersion made using the Arm field flume data were used to predict the wide flume downstream profiles. Figure 8 shows an example wide flume experimental downstream profile compared to a predicted downstream profile with $R^2_t = 0.7901$, indi cating there is a relatively good agreement between the recorded and predicted downstream data. Across the 139 wide flume trace records available, the mean R^2_t of predictions compared to recorded data is 0.7342 with a standard deviation of 0.0866. This shows the estimates of transverse and longitudinal dispersion for the wide flume based on data collected from theArmfield flume to be a good approximation.

Figure 9 and Figure 10 simplify the wide flume data shown in Figure 8 by taking the mean concen tration levels through time and transverse position respectively. Figure 9 shows that in this case the transverse variation of the solute has been slightly under-predicted ($R^2_t = 0.9519$) and that the estimate of D_y is fairly good. Figure 10 shows that the value of estimated D_x is reasonable, but that the longitudinal dispersion coefficient is more of an over-estimate (R^2_t = 0.8348) than D_y is an under-estimate. Both the under-estimate of D_y and the over-estimate of D_x are seen (although not presented) across the dataset. The over-estimation is not unexpected given that in the Armfield flume the shear layers at the side walls will generate turbulence and cause additional mixing. There is also the potential that the assumption of wall friction being similar to drag caused by the stems is incorrect, resulting in a velocity increase at the wall that increases differential longitudinal advection and therefore overall longitudinal dispersion. The consistent over-estimation of D_x and underestimation of D_y may be explained by the $D_x \propto 1/D_y$ relationship of (2), with therefore any error in D_x affecting the calculated value of D_y .Although the scale of additional processes affecting D_x is unknown, it might be reasonable to assume a reduction in D_x to compensate for the additional processes in future work and form 'better' estimates of D_x and D_y . Regardless, both the estimates of D_x and D_y for the wide flume are the correct order of magnitude and are fitfor-purpose for predicting mixing coefficients in a more realistic system.Transverse and longitudinal dispersion together may be estimated from: experiments conducted in a laboratory system designed to collect cross-sectionally well-mixed longitudinal dispersion data using traditional point solute tracing; CFD generated profiles of transverse variation of longitudinal velocity; and transverse dispersion calculated using Chikwendu (1986). 4 DISCUSSION Assuming the μ_t/S_{ct} estimate of dispersion was valid, the values of isotropic dispersion that would be calculated for the k − ε, k −ω, and RSM model results in Figure 6 respectively would be 2.2372×10^{-6} ,

Figure 9. A mean representation through time of the data

presented Figure 8. R^2_t = 0.9519.

Figure 10. A mean representation across transverse position

of the data presented Figure 8. R^2_t = 0.8348.

2.7278×10^{-7} and 2.1508×10^{-6} $m^2 s^{-1}$, assuming a

Schmidt number of 1. It is well known that S_{ct} is often

used as a calibration factor, but for any reasonable value around 1, the turbulence model estimated values of dispersion are still far smaller than the values of $D_x = 5.2864 \times 10^{-4}$ and $D_y = 1.2065 \times 10^{-5}$ $m^2 s^{-1}$ for the same configuration (obtained from the results in the previous section). This also suggests that the turbulence models may be under-estimating turbulence, similar to how the CFD generated profiles are less rounded than the experimental velocity profile of West (2016) velocity profiles (Fig. 5).

More complex modelling techniques such as Direct Numerical Simulation (DNS), Large Eddy Simulation (LES), and Lattice-Boltzmann Method (LBM) can potentially be used to produce better numerical results, e.g. Gac (2014), although this may come at great computational cost. It is clear however that CFD remains useful for predicting bulk flow characteristics, even in stem-scale vegetation, and that the marriage of experimental and computational results can be a powerful tool.

The joint project that has funded this research is attempting to build validated CFD modelling tools to help ensure that future pond designs meet water quality and ecosystem services objectives for both current legislation and the increasingly stringent EU regulatory framework anticipated over the next decade. The abil

West, P. (2016). Quantifying solute mixing and flow fields

in low velocity emergent real vegetation. Ph. D. thesis,

University of Warwick.

West, P., J. Hart, I. Guymer, & V. Stovin (2016). Devel

opment of a laboratory system and 2D routing analysis

to determine solute mixing within aquatic vegetation. In P. Rowinski and A. Marion (Eds.), Hydrodynamic and mass transport at freshwater aquatic interfaces. Springer.
Young, P., A. Jakeman, & R. McMurtrie (1980). An instrumental variable method for model order identification. Automatica 16(3), 281–294. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

SLIM: A model for the land-sea continuum and beyond

E. Deleersnijder

Université catholique de Louvain, Louvain-la-Neuve, Belgium

Delft University of Technology, Delft, The Netherlands

S. Blaise, P. Delandmeter, T. Fichefet, E. Hanert, J. Lambrechts, Y. Le Bars, V. Legat, J. Naithani,

C. Pham Van, J.-F. Remacle, S. Soares-Frazao, C. Thomas, V. Vallaeys & D. Vincent

Université catholique de Louvain, Louvain-la-Neuve, Belgium

T. Hoitink & M. Sassi

Wageningen University, Wageningen, The Netherlands

V. Dehant & O. Karatekin

Royal Observatory of Belgium, Brussels, Belgium

E. Wolanski

James Cook University, Townsville, Australia

ABSTRACT

The S econd-generation Louvain-la-Neuve Ice-ocean M odel (SLIM, www.climate.be/slim_flyer) deals with the equations governing sea-ice, geophysical, envi ronmental and groundwater phenomena by means of the (discontinuous Galerkin) finite element method on 1D, 2D or 3D unstructured meshes. To take advantage of state-of-the-art developments, SLIM is also being interfaced with existing tools (often based on radically different numerical methods), such as the well-known and widely used G eneral Ocean Turbulence Model (www.gotm.net, GOTM). The post-processing of the results is achieved with the help of usual statistical and computer graphics methods. Other techniques are also resorted to, such as tracer and timescale meth ods derived from CART (C onstituent-oriented Age and R esidence time Theory, www.climate.be/cart) or network science tools (sites.uclouvain.be/networks) (Thomas et al. 2014).The hydrodynamics simulated by the aforementioned finite element model can be intro duced into a number of SLIM-based environmental modules, which are capable of representing sediment transport (Delandmeter et al. 2015), as well as the fate of some classes of contaminants, namely microbiolog ical pollutants (de Brauwere et al. 2014), endocrine disrupting compounds, heavy metals (Elskens et al.

2014) or radionuclides. A simple ecological model is being developed, whose aim is to simulate the evo lution of various species of phytoand zoo-plankton (Naithani et al. 2016).

SLIM has been applied successfully to a wide vari ety of standard, idealised test cases for geophysical and environmental fluid flows — including atmospheric ones (Blaise et al. 2015). It was seen that space-time mesh adaptivity pays off. Realistic problems were or are also dealt with, in particular, the application of

Camus, P., M. Menéndez, F. J. Mendez, C. Izaguirre,

A. Espejo, V. Cánovas, J. Pérez, A. Rueda, I. J. Losada,

& R. Medina (2014). A weather-type statistical down

scaling framework for ocean wave climate. Journal of

Geophysical Research: Oceans 119(11), 7389–7405.

Cronshey, R. & U. S. S. C. S. E. Division (1986). Urban

hydrology for small watersheds. U.S. Dept. ofAgriculture,

Soil Conservation Service, Engineering Division.

Done, J., C. A. Davis, & M. Weisman (2004). The next

generation of NWP: explicit forecasts of convection

using the weather research and forecasting (WRF) model.

Atmospheric Science Letters 5(6), 110–117.

Fernández Bethencourt, J. D., P. Delgado Melián, J. López

García, & M. Rodríguez Pallarés (2009). Guía metodológ

ica para el cálculo de caudales de avenida en la isla de

Tenerife. In Jornadas de Ingeniería del Agua, Madrid,

pp. 1–10. Herrera, S., J. M. Gutiérrez, R. Ancell, M. R. Pons, M. D. Frías, & J. Fernández (2012). Development and analysis of a 50-year high-resolution daily gridded precipitation dataset over Spain (Spain02). International Journal of Climatology. Kalnay, E., M. Kanamitsu, & R. Kistler (1996). The NCEP/NCAR 40-year reanalysis project. Bulletin of the American Meteorological Society 77(3), 437–471. Menéndez, M., M. García-Díez, L. Fita, J. Fernández, F. J. Méndez, & J. M. Gutiérrez (2013). High-resolution sea wind hindcasts over the Mediterranean area. Climate Dynamics 42(7-8), 1857–1872. Skamarock, W. C., J. B. Klemp, J. Dudhia, D. O. Gill, D. M. Barker, W. Wang, & J. G. Powers (2005). A Description of the Advanced Research WRF Version 2. Technical report.

(Eds.) © 2016 Taylor & Francis Group, London, ISBN
978-1-138-02977-4

Application of artificial neural networks in meteorological drought forecasting using Standard Precipitation Index (SPI)

S. Golian & P. Yavari

Civil Engineering Department, Shahrood University of Technology, Shahrood, Iran

H. Ruigar

Shahrood University of Technology, Shahrood, Iran

ABSTRACT: Drought is one of the natural and climatic disasters and causes abundant damages to human life and natural ecosystems worldwide every year. This study aims to compare the results of meteorological drought forecasting using non-linear auto regressive (NARX) neural network model with those of commonly used multi-layer Perceptron (MLP) network for Shahrood climate station, located in North-East part of Iran. Different combination of input-output sets with various variables as inputs and SPI with different lag-time as output were tested to determine which combination has the best performance in prediction of future droughts. SPI drought index with 3 and 6 months aggregation period were applied in this study. In general, both MLP and NARX networks had satisfactory performance in prediction of SPI index, but NARX has slightly better performance compared to MLP network for all the models. Also predicting SPI6 resulted in better performance compared to SPI3. Results showed that for both models, using temperature, precipitation, humidity, SPI3, SPI6 and SPI12 as input variables and SPI6 with one month lag as output has better performance compared to other

input-output combination sets. The optimum calculated performance criteria were CNS (Nash-Sutcliff) = 0.954

and 0.962, R (correlation coefficient) = 0.989 and 0.986, MAE (mean of absolute error) = 0.033 and 0.036,

RMSE (root mean square error) = 0.041 and 0.043 for training and test periods, respectively for NARX network.

1 INTRODUCTION

Human kind has always faced with some kind of natu

ral hazards such as earthquake, volcano, flood, etc.

some of them produced from geological and geo

morphological activities, but others including storms,

droughts, torrential rainfall and lightning are produced

from climatic processes. Their intensity and frequency

are affected by geographical location.There, drought is

the most important natural catastrophe which is asso

ciated with decrease in rainfall and soil moisture and

increase in temperature and wind velocity. This event

has occurred and affected more human societies in

the last decade (The International Disaster Database,

2009). Agriculture sector would suffer more from

drought events because of its direct dependence on

water. For example, the Australian government spent

around 925 million dollars to catch up the drought

damages in 1970–1984. Furthermore the economic

loss in South Africa added up 2.5 billion dollars in

the middle of 70 decade (The International Disaster

Database, 2009). Drought prediction as a strategy to deal with this natural disaster and reduce the dam

aging effects is necessary and in recent years has

concerned the attention of scientists in climatologi

cal and agricultural sectors and various activities has

been made in order to monitor and predict these events precisely. Artificial intelligence algorithms are one of the widely-used methods drought forecasting. There are some indices to assess drought condition, e.g. standard precipitation index (SPI) is one of the most widely used indicators for assessing drought severity. In calculating SPI, the time scale can be considered for drought monitoring (McKee et al., 1993). Many studies have been done using the Standardized Precipitation Index for classifying the severity of drought, worldwide, e.g. in Iran (Golian et al. 2015), in Turkey Bancali et al. (2008), in Africa (Belayneh et al, 2013) and in USA (Lu Liu et al. 2013). SPI has shown accurate results, especially in drought detection (Guttmn et al., 1999) and could detect droughts at least one month earlier compared to other indices (Hayes et al., 1999). In this paper, we deliberated the application of SPI to study the drought events at Shahrood, North-East of Iran. We aim to apply artificial neural networks (ANN) as a forecasting tool for drought management at study region. Morid et al. (2007) used two drought indices including SPI and EDI (effective drought index) and applied ANN to predict drought occurrence in Tehran, Iran. In their study, EDI and SPI were set as output and various combinations of precipitation and previous Southern Oscillation Index (SOI) and North Atlantic Oscillation (NAO) were applied as model inputs. Results

showed that using previous drought index as model

input would increase the performance of the models for

drought forecasting. Karamooz et al. (2009) combined

the standard precipitation index, Palmer drought sever

ity index and water surface supply index to develop a

comprehensive combined drought index (HDI). HDI

was then used by a probabilistic neural network and a multilayer Perceptron network to forecast drought in 'Gavkhooni/Zayanderood' basin in central part of Iran. Results showed that both models have acceptable ability for drought forecasting.

Belayneh et al. (2013) compared the performance of five data driven models for forecasting long-term (6 and 12 months lead time) drought conditions at Awash River Basin, Ethiopia. The SPI with 12 and 24 aggregation period (SPI 12 and SPI 24) was forecasted using ARIMA, ANN and support vector regression (SVR) methods. Wavelet transforms were also used to pre-process the inputs for ANN and SVR mod els to form WA-ANN and WA-SVR models. Results indicated that the coupled wavelet neural network (WA-ANN) models performed better compared to other methods in forecasting SPI 12 and SPI 24 values with 6 and 12 months lead times.

The aim of this study was to compare the results of forecasting meteorological drought index, SPI index using non-linear auto regressive (NARX) neural net work model with those of commonly-used multi-layer perceptron (MLP) network. Different combination of input-output sets with various variables as inputs and SPI with different lag-time as output were tested to determine which combination has the best perfor mance in prediction of future droughts. For this aim,

SPI drought index with 3 and 6 months aggregation

periods were used in this study.

Figure 1. Geographical location of Shahrood synoptic

station. Table 1. Features of Shahrood synoptic station. Precipitation (mm) Longitude (E) Latitude (N) Height (m) max min Average 54 ∘ 57 ′ 36 ∘ 25 ′ 1345.3 299.6 68.6 157.6 2 STUDY AREA The study area is Shahrood city. The southern part has warm and dry weather in the vicinity of desert. The central division and the eastern zones benefit by temperate semi-arid climate and the northern part adjoins by mountainous area. The average annual temperature and precipitation is 14 degrees Celsius and 157.6 millimeter, respectively. The average height from sea level is about 1380 meter. Monthly Precipitation depth (mm), temperature (degree Celsius) and relative humidity data for period 1990–2012 of Shahrood synoptic station was used to calculate SPI with 3 and 6 months aggregation periods (Figure 1) as inputs to models. Table 1 contains some statistics for this station. 3 MATERIALS AND METHODS 3.1 Standardized Precipitation Index (SPI) Standardized Precipitation Index (SPI) is presented to quantify the lack of precipitation for several different time scales based on precipitation data. In its calculation, SPI uses only rainfall data and is capable to account for a wide range of time scales from one month to several years. For more details readers are referred to McKee, et al, 1993 and Hayes, 2001. Table 2 contains the SPI time-scale classification: 3.2 Artificial Neural Networks (ANN) Artificial intelligence methods have the ability to give better performance through handling nonlinearity and other complexities in modeling of the time series. From these categories, Artificial Neural Networks (ANNs) have significant flexibilities in data modeling (e.g. Zhang and Dong, 2001; Gan et al., 2009). In this study, Feed-Forward Multi-layer network which is also called Multilayer Perceptron network (MLP) was used for drought forecasting. In MLP, neurons of each layer are connected to all of neurons of the previous layer. For each neuron, the inputs are calculated as follow: Where net n i is the input value of the ith neuron in nth layer, w n ji is the connecting weight between ith neuron in nth layer and jth neuron in the (n − 1)th layer and,

O n−1 j is the output of the jth neuron at (n − 1)th layer, and m is the number of neurons in the (n − 1)th layer (Ghalkhani et al, 2013).

There are some learning algorithms for training neural networks, e.g. back-propagation, conjugate gra dient and Levenberg-Marquardt (LM). From 1993 onward, Levenberg − Marquardt algorithm is recog nized as the fastest method for training neural networks and hence, we used it as the training algorithm in our study. Also, sigmoid (sigm) and linear (purlin) trans fer functions were used in the hidden layer and output layers, respectively.

Also there are no systematic rules to obtain the num ber neurons of the hidden layer neurons. Small number of neurons for the network might create low fitness while large number of neurons might create high fit ness (Dawson & Wilby, 1998). The number of neurons in the hidden layer was determined by trial and error ranging from 1 to 20 neurons. 75% of data were used for training, 10% for validation and 15% for testing stage.

3.3 Autoregressive networks

NARX is a recurrent dynamic neural network with feedback connections enclosing several layers of the network. NARX model is based on linear auto regres sive exogenous model (ARX) which is commonly used for the task of prediction. The model can be stated

mathematically as:

Table 2. Drought classification upon SPI (Hayes, 2001).

SPI	Drought level
2 and upper	Severe wet season
1.5–1.99	Extreme wet season
1–1.49	Mild wet season
-0.99–0.99	Standard
-1.49–-1	Mild drought
-1.99–-1.5	Extreme drought
-2 and lower	Severe drought

Table 3. Input models for prediction.

Prediction

Model	Input	Outputs
1	SPI 3 (t)	SPI 3 with ((t+1), (t+3), (t+6), (t+12)) lags
	SPI 6 (t)	SPI 6 with ((t+1), (t+3), (t+6), (t+12)) lags
2	SPI 3(t) , SPI 6(t) , SPI 12(t)	SPI 3 with ((t+1), (t+3), (t+6), (t+12)) lags
	SPI 3(t) , SPI 6(t) , SPI 12(t)	SPI 6 with ((t+1), (t+3), (t+6), (t+12)) lags
3	SPI 3(t) , Rainfall , Humidity , Temperature	SPI 3 with ((t+1), (t+3), (t+6), (t+12)) lags
	SPI 6(t) , Rainfall , Humidity , Temperature	SPI 6 with ((t+1), (t+3), (t+6), (t+12)) lags
4	SPI 3(t) , SPI 6(t) , SPI 12(t), Rainfall , Humidity , Temperature	SPI 3 with ((t+1), (t+3), (t+6), (t+12)) lags
	SPI 3(t) , SPI 6(t) , SPI 12(t), Rainfall , Humidity , Temperature	SPI 6 with ((t+1), (t+3), (t+6), (t+12)) lags

Where y is the output, x is the input, and d x and d y are the delay of the inputs and outputs, respectively. To develop an MLP and NARX ANN models, different combinations of input variables, i.e. relative humidity, temperature, precipitation and standardized precipitation index (SPI 3 and SPI 6) were used together with SPI3 and SPI6 with various lag-time as output. Table 3 contains four ANN prediction models used in this study. In order to determine the performance of ANN's, some criteria including correlation coefficient (R), root mean square error (RMSE), Nash Sutcliff criterion (CNS) and mean of absolute error (MAE) were used. 3.4 Model performance To determine the accuracy of models, appropriate criteria should be used. Models' convergence is also controlled by these indices. In this case, observed and predicted SPI values were used to calculate three performance statistics, i.e. RMSE (Root Mean Square Error) MAE (Mean absolute error), CNS (NashSutcliffe coefficient) and correlation coefficient (R) as follow: The MAE index is a non-negative index and provides the average deviation of the predicted values from observations. On the other hand, RMSE is more sensitive to simulation errors compared to MAE. The mean absolute error is a common measure of forecast error in time series analysis (Hamilton 1994). The Nash-Sutcliffe index has values ranging from $-\infty$ to 1 with CNS = 1 being the optimal value.Values between 0 and 1 are generally viewed as acceptable levels of performance, whereas values <0 indicates that the mean observed value is a better predictor than the simulated value and indicates unacceptable performance (Moriasi et al. 2007).

Table 4. Most severe droughts occurred in SPI3 and SPI6.

	First event			Second event			Third event		
SPI	Start date	Drought duration (month)	Lowest index	Start date	Drought duration (month)	Lowest index	Start date	Drought duration (month)	Lowest index
3	Oct-96	6	-3.53	Apr-99	5	-2.89	Feb-07	7	-2.19
6	Nov-96	11	-2.86	Mar-99	6	-2.57	Feb-07	9	-2.15

Table 5. The results of prediction models using MLP network.

4 RESULTS AND DISCUSSION

Table 4 contains the first three most severe drought events calculated for Shahrood synoptic station for the period 1992-2011. The SPI =-0.5 was considered as a threshold for drought onset (Golian et al., 2014).

For the NARX model, 10 neurons at the hidden layer and two delays for both inputs and outputs were derived to exhibit optimum behavior.Tables 5 and 6 present the performance criteria for prediction of SPI3 and SPI6 drought indices. At these tables, the model with best performance are shown with another color.

Tables 5 summarizes the performance of the devel oped MLP models for training and test datasets for interested models. The statistics presented in Table 5 indicate that the 4th model with SPI3, SPI6, SPI12, Rainfall, Humidity and Temperature as inputs and SPI6 with 1 month lag as the output, has performed bet ter compared to other models.Also, the values of R and CNS for this model are 0.989 and 0.956, respectively, in the training phase and 0.986 and 0.961, respectively, in the test phase. This emphasizes the importance of rainfall and a drought index itself as inputs of MLP model to result in accurate forecasting. Table 6 illustrates the performance of NARX prediction models with different inputs. Table 6 also indicates that the 4th model with SPI3, SPI6, SPI12, Rainfall, Humidity and Temperature as inputs and SPI6 with 1 month lag as an output, has performed better compared to other models.Also, the values of R and CNS for this model are 0.989 and 0.954, respectively, in the training phase and 0.986 and 0.962, respectively, in the test phase. In general, increasing the lead-time resulted in decreasing prediction accuracy. The best result among all models and both networks obtained for prediction of SPI 6 with 1 month delay. Also, the NARX model performed slightly better compared to the MLP. The scatter diagram of this state for all test and train data are presented in Figure (2). 5 CONCLUSION Drought prediction is one the strategies to manage this natural phenomenon and reduce its negative effects. One of the most widely used indicator to classify the drought severity is the standard precipitation index (SPI). One of the methods in drought forecasting is artificial intelligence algorithms, e.g. artificial neural networks (ANNs). This study aimed to compare the

Table 6. The results of prediction models using NARX network.

Figure 2. Scatter plots of the best model in SPI prediction.

results of meteorological drought forecasting with SPI index using different variables as inputs of the ANN models. In this paper all data including calculated SPI, temperature, humidity and precipitation were normal ized between 0 and 1 values to predict SPI, using neural network for Shahrood synoptic station, Iran. In this study, we used various combinations of input and out puts. Using climate variables combined with drought indices improved the neural network results in drought prediction. However the use of climate variables or drought indices alone as the input parameters of the ANN models showed less accurate results compared to the input combination with both meteorological and SPI variables (in both MLP and NARX, models 3 and 4 had better results compared to models 1 and 2). As

Ghalkhani, Hossein, Saeed Golian, Bahram Saghafian, Ashkan Farokhnia, and Asaad Shamseldin. 2013. “Appli cation of Surrogate Artificial Intelligent Models for Real Time Flood Routing.” Water and Environment Journal 27(4): 535–48.

Golian. S, Mazdiyasni. O, AghaKouchak. A., (2015), Trends in meteorological and agricultural droughts in Iran, Theor

Appl Climatol., 119, 679-688.

Guttmn, B, N. 1999. Comparing the Drought Index and the Standardized Precipitation Index. J of American Water Resources Association: 34(1):113-121.

Hamilton JD (1994) Time series analysis. Princeton Univer sity Press, Princeton.

Hayes, M. J. 2001. Drought Indices, National Drought Mitigation Center, Noaa.

Hayes, M. J., Svoboda, M.D., Wilhite, D.A. and Van yarkho, O. 1999. Monitoring the 1996 DroughtUsing the Standardized precipitation Index. Bulletin of American Meteorological Society. 80(3): 429-438.

Haykin, S. (1994). Neural networks: a comprehensive foun dation. MacMillan Publishing Company.

Horne, B.G. and C.L. Giles, 1995. An Experimental Compar ison of Recurrent Neural Network. 7th Edn. MIT Press, pp: 697. clgiles.ist.psu.edu/papers/NIPS94.rnn.comp.pdf.

Hsu, KL, Gupta, HV, sorooshian, S. (1995) Artificial Neu ral Network modeling of rainfall-runoff modeling. Water Resour. Res., 31, 2517-2530.

Jae Won Kwak, Min Soo Kyoung, Duck Gil Kim, Hung Soo Kim. Impact of Climate Change on SPI and SAD curve. World Environmental and Water Resources Congress (2011).

Karamouz M.; Rasouli K.; and Nazif S. (2009). Devel opment of a Hybrid Index for Drought Prediction)

Case Study. (Journal Of Heydrologic Engineering ASCE,

14(6), 617-627.

Leontaritis I.J. and Billings S.A.. 1985, "Input-output para

metric models for non-linear systems. Part I: deterministic

non-linear systems". Int'l J of Control 41:303-328.

Lu Liu; Yang Hong; Jonathan Looper; Rachel Riley; Bin

Yong;Zengxin Zhang; James Hocker; and Mark Shafer.

Climatological Drought Analyses and Projection Using

SPI and PDSI (Case Study of the Arkansas Red River

Basin). Journal Of Heydrologic Engineering ©ASCE July

(2013). McKee, T. B., Doesken, N. J., and Kleist, J. (1993). "The relationship of drought frequency and durationto time scales." Proceedings of the 8th Conference on Applied Climatology, 179-184. Mckee,T.B., N.J. Doesken, and Kleist, J., 1995. Drought monitoring with multiple time scales. Ninth Conference on Applied Climatology, American Meteorological Society, Jan 15-20, 1995, Dallas TX, pp. 233-236. Moriasi DN, Arnold JG, Van Liew MW, Binger RL, Harmel RD, Veith T (2007) Model evaluation guidelines for systematic quantification of accuracy in watershed simulations. TransASABE 50(3):885-900. Morid, S, Smakhtin, V & Bagherzadeh, K (2007). Drought forecasting using artificial neural networks and time series of drought indices, International Journal of Climatology, Vol. 27. Najafi B. and Tarazkar M.H., (2006) "Applications of Artificial neural networks to predict pistachio export of Iran" Business bulletin, pp.191-214 (In Persian). Sajikumar, N.,Thandaveswara, B. S., 1999. Non Liner rainfall runoff Model using artificial neural network, Journal of Hydrology, 216, pp 32-35. Smith, J, Eli, RB, neural Network models of rainfall-runoff process. J. Water Resour. Plng. Mgmt., 4, 232-239. Tedesco M, Pulliainnen J, Takala M, Halikainen M and Pampaloni P 2003 Artificial neural network-based techniques for the retrieval of SWE and snow depth from SSM/I data; Remote Sens. Environ. 90, 76-85. The OFDA/CRED International Disaster Database, (2009). www.em-dat.net. Tokar, AS, Johnson, PA, (1999) Rainfall-runoff modeling using artificial neural networks. J. Hydr. Eng., 4, 232–239. Tokar, AS, Markus, M, (2000) Precipitation-runoff modeling using artificial neural networks and conceptual models. J. Hydr. Eng., 5, 156–161. Zhang B L and Dong Z Y 2001 An adaptive neural-wavelet model for short term load forecasting; Electer. POW. Syst. Res. 59 121–129. Zhang, B. Govindaraju, RS, (2003) Geomorphology-based artificial neural networks for estimation of direct runoff over watersheds. J. Hydrol., 273, 18–34. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Accuracy assessment of ISI-MIP and FAO hydrological modelling results

in the Upper Indus Basin

Asif Khan

Department of Engineering, University of Cambridge, UK

Department of Engineering, University of Engineering and Technology, Peshawar, Pakistan

Internationational Institute for Applied System Analysis, Laxenburg, Austria

Mujahid Khan

Department of Engineering, University of Engineering and Technology, Peshawar, Pakistan

ABSTRACT: Millions of people rely on river water originating from snow and ice melt in the Hindukush

Karakoram-Himalayan (HKH). One such basin is the Upper Indus Basin (UIB), where snow and glacier melt

contribution is more than 80%, therefore is highly susceptible to global warming and climate change. Accuracy

of available hydro-climatic studies' results are vital for future precise policy making and sustainable water

resource development. Therefore, this research evaluates accuracy of various ISI-MIP and FAO hydro-climatic

studies results, during 1985–1998 and 1961–1990 respectively, for six sub-basins of the UIB. This research

evaluates accuracy of bias corrected five GCMs' precipitation data sets, input of ISI-MIP hydrological models,

and CRU data, input of FAO hydrological model, all based on basin-wide mass balance assessment. First the input

precipitation data sets have been evaluated followed by comparison of modelled and measured flows. Basin-wide

mass balance assessment show that all precipitation data sets used in ISI-MIP and FAO hydrological models

significantly underestimate precipitation in the UIB, particularly in the Karakoram sub-basins. All ISI-MIP (6)

and FAO hydrological models provide consistent but significantly low modelled flows (<50%) as compared

to the measured records in all sub-basins, except for the Kharmong basin. FAO water scarcity shows severe

water scarce conditions in the UIB. FAO and ISI-MIP under-estimated modelled flows (and water scarcity) are

artefacts of use of under-estimated precipitation data use. This study shows that results of ISI-MIP and FAO

are not true representative of hydro-climatic conditions in the UIB, therefore cannot be used in precise and

accurate policy making and water resource management.

1 BACKGROUND AND INTRODUCTION

Global hydro-climatic studies are needed for under

standing climate change impact on global, continental

and regional scale. Global hydrological modelling

may compromise regional and local hydro-climatic

conditions, particularly in complex mountain basins.

Warming and changing climate together with popula

tion growth may significantly increase stress on water resources in alpine regions. Therefore for precise and accurate policy making and sustainable water resource development, global hydro-climatic studies need to be validated at regional and local scale to support results and or to improve the adopted models. However, no or limited global studies data sets, results and conclusion have been validated at regional scale.

Therefore, this study evaluates accuracy of results of two well-known global hydro-climatic studies: i) Inter-Sectoral Impact Model Intercomparison Project (ISI-MIP; Warszawski et al., 2014), ii) FAO hydro logical modelling results and water-scarcity, all in

the selected sub-basins of the Upper Indus Basin. Accuracy of six hydrological models from ISI-MIP: i) H08, ii) VIC, iii) WaterGAP, iv) WBM, v) MPI-HM, vi) PCR-GLOBWB, and FAO's GLOBWAT hydrological model has been assessed in the Upper Indus Basin, its sub-basins and for the entire Indus Basin. Accuracy of input precipitation data has also been assessed. Unfortunately, both ISI-MIP and FAO modelled flows have been found to be far under-estimated compared to measured flows, and cannot be used in precise and accurate policy making and water resource development, therefore need revisit and improvement. 2 THE STUDY AREA In Pakistan, almost all agricultural land is dependent on irrigation, and irrigated agriculture yields about 90% of the agricultural production per annum (Afzal 1996), which is about 24% of total gross domestic product (GDP) (Muneer and Asif, 2007; Piracha and Majeed, 2011). Some 70% of this irrigation is provided

by the Indus Basin Irrigation Network, which is one of the world's largest irrigation networks, irrigating 17 million hectares (MHa) in a total of a 24 MHa cul tivable command area (Wescoat et al., 2000; Kahlown et al., 2007). The Indus River annual average flow is about 4 million m^3, and most of its flow (60–80%) originates from seasonal snow and glacier melt in the HKH (see e.g., Lutz et al., 2014; Mukhopadhyay and Khan, 2014). Thus, change in temperature or precip itation can significantly influence downstream water supplies.

Therefore, the current study are is the Upper Indus River Basin, which has a drainage area of about 172,173 km^2, which is tapped by the Tarbela Dam (see delineation details in Khan et al., 2014). The Tarbela Dam is the main source of fresh water for irrigation and power generation in Pakistan. It pro vides about 60% of irrigation water supplies, and more than 25% of power generation (Archer 2003).

The UIB originates from the Hindukush-Karakoram Himalayan (HKH) and Tibetan Plateau (TP) region (see Figure 3). The estimated area of the UIB at Tar bela Dam is 172,173 km^2 (Khan et al., 2014).There are six main sub-basins of the UIB: i) Gilgit, ii) Hunza, iii) Shigar, iv) Shyok, v) Kharmong, and vi) Astore (see Figure 3). Although detailed analysis is limited to the UIB, however water stress and scarcity analysis for the entire Indus Basin have also been carried out.

3 DATA AND METHODS

ISI-MIP provides output results (flow data) of hydro logical models based on five Global Circulation Mod els (GCMs) bias corrected precipitation data sets. These GCMs are: a) GFDL-ESM2M, b) Had GEM2 ES, c) IPSL-CM5A-LR, d) MIROC-ESM-CHEM, and d) NorESM1-M. GCMs output precipitation data sets have been downscaled and bias corrected with (WFDEI) WATCH Forcing Data ERA Interim (Weeden et al., 2011) data, using method explained in Hemple et al. (2013), and are available on a 0.5 ∘ grid. Both monthly precipitation for all GCMs, WFDEI and monthly modelled flow data of all hydrological models have been acquired for a base-period of 1985–1998 (consistent with the least available measured flow record acquired from WAPDA).

FAO hydrological model utilized Climate Research Unit (CRU) precipitation data. Therefore, FAO mod elled flows have been acquired for the period 1961–1990, while CRU Time Series (TS) v 3.22 gridded monthly precipitation data (Harris et al., 2014) are available on a 0.5 ∘ × 0.5 ∘ grid from 1901 to 2013. CRU data are based on climatic station precipitation data. Using the website: http://www.cru.uea.ac.uk/cru/data/hrg/ (accessed on 5 December, 2014), the data have been acquired, while for consistency with other datasets only 1979–2010 data have been extracted and

used in the current study.

Gauge stream flow datasets for the UIB sub

basins have been acquired from Water and Power Development Authority (WAPDA). However, for the entire Indus Basin long term average renewable water resources estimates have been acquired from Laghari et al. (2012) and Aquastat (2011–2012). 3.1 Basin-wide mass balance analysis Basin-wide mass balance analysis have been carried out using method explained in Reggiani and Reintjes (2014). However, in the current study average actual evapotranspiration losses have been estimated using data from ERA interim (Dee et al., 2011), FAO, MODIS (Mu et al., 2011), and data from Bastiaanssen et al. (2012). Net-glacier melt contributions to flows have been estimated using glacier mass balance studies (Kaab et al., 2012, 2015; Gardelle et al., 2012, 2013). As glacier mass balance studies are available only for the period 2000–2011, therefore mass balance analysis has also been carried out for nearly the same period (1999–2010). To check accuracy of precipitation data, a minimum threshold limit has been defined by adding measured flow, minimum actual evapotranspiration together with glacier net-mass balance flow contribution (negative in case of negative mass balance and vice versa) in a sub-basin. However, as actual evapotranspiration is significantly dependent on input precipitation data, therefore a maximum threshold limit has been defined by flow, potential evapotranspiration and glacier net-mass balance flow contribution in a sub-basin. Potential evapotranspiration datasets have been acquired for the same period, while average potential evaporation data from ERA interim, CRU and FAO has been adopted. Thus, any data below minimum threshold limit has been considered to be under-estimated precipitation data, while above maximum threshold limit as over-estimated precipitation data (see details in below section 4). 4 RESULTS AND DISCUSSION Initially, the accuracy of precipitation data sets have been assessed, followed by comparison of modelled flows data with measured flows, all for the UIB subbasins. Unfortunately, all the selected precipitation datasets have been found to be far below (3–5 times) than minimum threshold limit in all sub-basins (such as shown for the Hunza basin in Figure 1), except for the Kharmong, where precipitation datasets have been found to be above minimum threshold limit but below maximum threshold limit. Main cause of such under-estimation is use of limited valley based climate stations, which receive 5–10 folds less precipitation than higher altitudes, and cannot represent entire sub-basin's precipitation (see e.g., Immerzeel et al., 2015; Dahri et al., 2016). Precipitation in the Kharmong basin is reasonable because there are significant amount of climate stations in the south of the basin, and their recorded precipitation is far greater

Figure 1. Accuracy assessment of precipitation datasets in the Hunza basin, based on mass balance analysis. All data sets

are for the period 1999–2010. Lower and Upper Limits are minimum and maximum mass balance threshold limits described

in methods (see Section 3).

than valley-based stations in the UIB. Therefore, the interpolated precipitation in the Kharmong basin is entirely different from other sub-basins.

GCMs output precipitation data sets have been downscaled and bias corrected with WFDEI (Weeden et al., 2011) data, which under-estimate precipitation in sub-basins of the UIB, thus bias-corrected down scaled data also under-estimate precipitation in all sub-basins, except the Kharmong basin (see Figure 1).

Of the five GCMs precipitation datasets, Had GEM2 ES provides nearly average precipitation compared to average of all GCMs outputs. Therefore, modelled flows based Had GEM2-ES have been discussed in below paragraphs.

All hydrological models provide consistent results for various GCMs input data (see Figure 2). How ever, of the six hydrological models, two models:

MPI-HM and WBM provides far less modelled flows compared to the other models results and measured flows (Figure 2). This suggests that both MPI-HM and WBM are not suitable for the UIB. The remaining four models provide consistent results but far less than mea sured flows in all sub-basins, except for the Kharmong basin, where modelled and measured flow estimates are nearly the same (see Figure 2). Of the selected six hydrological models H08 and VIC are energy-based hydrological models, while all other remaining mod els are temperature-based, therefore require different input variables other than mutually used precipitation data (for details see Davie et al., 2013). Interestingly, both energyand temperature-based hydrological models (except MPI-HM and WBM models) provide nearly same consistent results, and could be due to use of same input precipitation data, although other input data and calibration parameters require further investigation. However, modelled flow data significantly underestimated renewable surface water in the UIB, and are not true representatives of this region. Main cause of modelled flow under-estimation is use of under-estimated precipitation. Most of the under-estimation can be noticed in the Gilgit, Hunza, Shigar and Shyok basins, where precipitation is significantly understimated (see Figure 2), while glacier mass balance is slighlty negative (Kaab et al., 2012, 2015). Thus, confirms that main cause of underestimated modelled flows is use of under-estimated precipitation data. In addition to ISI-MIP annual measured and modelled flow comparsion, monthly modelled and measured flows have also compared (see Figure 3). This figure shows that measured and modelled flow seasonal variability is not compatible in all sub-basins, and for all hydrological models, except for VIC model in the Kharmong basin (see Figure 3 a-g). Modelled flows have been over-estimated in spring-early summer and significantly under-estimated in summer months.

Figure 2. Performance of various hydrological models in the

UIB and its sub-basins, for Had GEM2-ES precipitation data.

All data sets are compared with measured flow during 1985–1998.

Figure 3. Monthly variation of modelled and measured flows in the UIB sub-basins, all for the period 1985-1998. Location

of the HKH mountain ranges are also shown. Glaciers are based on GLIMS data. Imagery source is: Esri, DigitalGlobe,

Earthstar Geographics, CNES/Airbus DS, USDA, USGS, AEX, Getmapping, Aerogrid, IGN, IGP, swiss topo, and the GIS

User Community.

All this suggests, that ISI-MIP historic flows are

significantly under-estimated, and cannot be used in

the study area without prior rectification. However,

flow data of ISI-MIP have been used in water-scarcity estimation by Schewe et al. (2014), which shows severe water scarcity conditions in the UIB and nearby other basins. Therefore results and conclusions of Schewe et al. (2014) are also biased, at least for the UIB,

Figure 4. FAO/GAEZ water-scarcity in various Asian basins (Hoogeveen et al., 2015), and monthly measured and modelled

flows in various sub-basins of the UIB. Locations of flow gauging stations at Tarbela Dam and Panjnad are also shown. Water

scarcity limits are irrigation evapotranspiration in percentage of available renewable water resource (<10% is low, 10–20% is

moderate, and >20% is high).

although need further investigation for other nearby

basins.

Similarly, GAEZ/FAO used GLOBWAT hydro

logical model to estimate renewable surface water resources at a fine resolution, using CRU CL 2.0 (10 min resolution) precipitation data (New et al., 2000), during 1961–1990. Water scarcity based on renewable water resources and evaporation during 1961–1990 is provided in Figure 4 (data obtained from http://www.fao.org/geonetwork). This figure shows an alarming situation in most of the Asian basins, where these basins have remained under severe water scarce condition during 1961-1990. However, the accuracy assessment results of CRU CL2.0 show similar underestimation of precipitation in the UIB as CRU TS 3.1 data (discussed above). Therefore, use of underestimated precipitation data resulted in significantly under-estimated modelled flows (see Figure 4), thus suggested severe water scarcity during 1961–1990.

The UIB contributes about 40–50% of total flows in the Indus Basin (based on river flows during 1985–2000 at Tarbela Dam and Panjnad station, shown in Figure 4), and this suggests an overall biased water scarcity estimates for the entire Indus Basin. The results of modelled flows for the whole Indus basin confirms this assertion, where modelled specific flows (about 5 mm/yr) are 18 times lower than measured specific flows (90 mm/yr), provided in Hoogeveen et al. (2015). The suggested water scarcity in the Indus basin may result in ill-informed and inappropriate policy making to exploit existing water resources, such as construction of new dams, therefore FAO/GAEZ biased water scarcity results need rectification and revisiting in the Asian basins, at least in the Indus basin. 5 CONCLUSIONS AND RECOMMENDATIONS i. Bias corrected precipitation datasets (and their derived modelled flows) produced by ISI-MIP significantly under-estimate precipitation (renewable flow). Data derived from ISI-MIP flows, such as water scarcity, drought and floods, and their forecasts are biased for the UIB, and for the entire Indus Basin, and cannot be used for water resource planning and management. ii. CRU, WFDEI and GCMs ouput precipitation data, used in FAO and ISI-MIP hydrological models, significantly under-estimate precipitation in the UIB, and need bias correction prior to use. iii. ISI-MIP energyand temperature-based four hydrological models (H08, VIC, PCR-GLOBWB,

and WaterGAP) provide consistent results. One plausible cause of this consistency could be use of same precipitation data, however influence of other input data sets and calibration parameters need further research.

iv. Significant under-estimated biased results of WBM and MPI-HM (used in ISI-MIP) cautions to avoid use of these models for the UIB, as well as suggests to select hydrological models with great care in mountain regions.

v. ISI-MIP and FAO historic water-stress and water scarcity are artefacts produced by under-estimated flows (caused by use of under-estimated precipita tion data). However other input datasets, calibra tion parameters and types of hydrological models need further investigation.

vi. This study also suggest to evaluate results of other global hydro-climatic studies in the Indus Basin, and to exploit additional potential reasons for under-estimated modelled flows.

Thus, it is intensively needed to revisit above men tioned hydro-climatic studies, and to carry out such analysis in nearby other HKH-TP basins. To improve future hydro-climatic studies, researchers are recom mended to avoid use of under-estimated precipitation and other biased calibration parameters. The cur rent study will help to improve future hydro-climatic studies for precise and accurate policy making and sus tainable water resource development in the UIB and nearby other HKH-TP basins.

Afzal, M. 1996. Managing Water Resources for Environmen tally Sustainable Irrigated Agriculture in Pakistan. The Pakistan Development Review, 35, 977–988.

Aquastat 2011–2012. Irrigation in Southern and Eastern Asia in figures FAO-Water Report 37. Italy, Rome: Food and Agricultural Organization of the United Nations.

Archer, D. 2003. Contrasting hydrological regimes in the upper Indus Basin. Journal of Hydrology, 274, 198–210.

Bastiaanssen, W. G. M., Cheema, M. J. M., Immerzeel, W. W., Miltenburg, I. J. & Pelgrum, H. 2012. Surface energy balance and actual evapotranspiration of the transbound ary Indus Basin estimated from satellite measurements and the ETLook model. Water Resources Research, 48,

n/a-n/a.

Dahri, Z. H., Ludwig, F., Moors, E., Ahmad, B., Khan, A. &

Kabat, P. 2016. An appraisal of precipitation distribution

in the high-altitude catchments of the Indus basin. Science

of The Total Environment, 548–549, 289–306.

Dee, D. P., Uppala, S. M., Simmons, A. J., Berrisford, P., Poli,

P., Kobayashi, S.,Andrae, U., Balmaseda, M.A., Balsamo,

G., Bauer, P., Bechtold, P., Beljaars, A. C. M., Van De

Berg, L., Bidlot, J., Bormann, N., Delsol, C., Dragani,

R., Fuentes, M., Geer, A. J., Haimberger, L., Healy, S.

B., Hersbach, H., Hőlm, E. V., Isaksen, L., Kållberg, P.,

Köhler, M., Matricardi, M., Mcnally, A. P., Monge-Sanz,

B. M., Morcrette, J. J., Park, B. K., Peubey, C., De Rosnay,

P., Tavolato, C., Thépaut, J. N. & Vitart, F. 2011. The ERA

Interim reanalysis: configuration and performance of the

data assimilation system. Quarterly Journal of the Royal

Meteorological Society, 137, 553–597. Gardelle, J., Berthier, E.,Arnaud,Y. & Kääb,A. 2013. Regionwide glacier mass balances over the Pamir-KarakoramHimalaya during 1999–2011. The Cryosphere, 7, 1263- 1286. Gardelle, J., Berthier, E., Arnaud, Y. 2012. Slight mass gain of Karakoram glaciers in the early 21st century. Nature Geosci., 5, 322–325. Harris, I., Jones, P. D., Osborn, T. J. & Lister, D. H. 2014. Updated high-resolution grids of monthly climatic observations – the CRU TS3.10 Dataset. International Journal of Climatology, 34, 623–642. Hempel, S., Frieler, K., Warszawski, L., Schewe, J. & Piontek, F. 2013. A trend-preserving bias correction – the ISI-MIP approach. Earth Syst. Dynam., 4, 219–236. Hoogeveen, J., Faurès, J. M., Peiser, L., Burke, J. & Van De Giesen, N. 2015. GlobWat – a global water balance model to assess water use in irrigated agriculture. Hydrol. Earth Syst. Sci. Discuss., 12, 801–838. Immerzeel, W. W., Wanders, N., Lutz, A. F., Shea, J. M. & Bierkens, M. F. P. 2015. Reconciling high-altitude precipitation in the upper Indus basin with glacier mass balances and runoff. Hydrol. Earth Syst. Sci., 19, 4673–4687. Kääb, A., Berthier, E. N., Christopher, Gardelle, J. & Arnaud, Y. 2012. Contrasting patterns of early twenty-first-century glacier mass change in the Himalayas. Nature, 488, 495–498. Kääb, A., Treichler, D., Nuth, C. & Berthier, E. 2015. Brief Communication: Contending estimates of 2003– 2008 glacier mass balance over the Pamir–Karakoram– Himalaya. The Cryosphere, 9, 557–564. Kahlown, M. A. R., Abdur Zubair, Muhammad Kemper, W. Doral 2007. Water use efficiency and economic feasibility of growing rice and wheat with sprinkler irrigation in the Indus Basin of Pakistan. Agricultural Water Management, 87, 292–298. Khan, A., Richards, K. S., Parker, G. T., Mcrobie, A. & Mukhopadhyay, B. 2014. How large is the Upper Indus Basin? The pitfalls of auto-delineation using DEMs. Journal of Hydrology, 509, 442–453. Laghari, A. N., D.Vanham, And W.Rauch 2011. The Indus basin in the framework of current and future water resources management. Hydrology and Earth System Sciences, 8, 2263–2288. Lutz, A. F., Immerzeel, W. W., Shrestha, A. B. & Bierkens, M. F. P. 2014. Consistent increase in High Asia's runoff due to increasing glacier melt and precipitation. Nature Clim. Change, advance online publication. Mu, Q., Zhao, M. & Running, S. W. 2011. Improvements to a MODIS global terrestrial evapotranspiration algorithm. Remote Sensing of Environment, 115, 1781–1800. Mukhopadhyay, B. & Khan, A. 2014. A quantitative assessment of the genetic sources of the hydrologic flow regimes in Upper Indus Basin and its significance in a changing climate. Journal of Hydrology, 509, 549–572. Muneer, T. & Asif, M. 2007. Prospects for secure and sustainable electricity supply for Pakistan. Renewable and Sustainable Energy Reviews, 11, 654–671. Piracha, A. & Majeed, Z. 2011. Water Use in Pakistan's Agricultural Sector: Water Conservation under the Changed Climatic Conditions. Water Resources and Arid Environments, 1, 170–179. Reggiani, P. & Rientjes, T. H. M. 2014. A reflection on the long-term water balance of the Upper Indus Basin. Hydrology Research, 1–17. Schewe, J., Heinke, J., Gerten, D., Haddeland, I., Arnell, N. W., Clark, D. B., Dankers, R., Eisner, S., Fekete, B. M., Colón-González, F. J., Gosling, S. N., Kim, H., Liu, X., Masaki, Y., Portmann, F. T., Satoh, Y., Stacke, T.,

Tang, Q., Wada, Y., Wisser, D., Albrecht, T., Frieler, K., Piontek, F., Warszawski, L. & Kabat, P. 2014. Multi

model assessment of water scarcity under climate change.

Proceedings of the National Academy of Sciences, 111,

3245–3250.

Warszawski, L., Frieler, K., Huber, V., Piontek, F., Serdeczny,

O. & Schewe, J. 2014. The Inter-Sectoral Impact Model

Intercomparison Project (ISI-MIP): Project framework.

Proceedings of the National Academy of Sciences, 111,

3228–3232. Weedon, G. P., Gomes, S., Viterbo, P., Shuttleworth, W. J., Blyth, E., Österle, H., Adam, J. C., Bellouin, N., Boucher, O. & Best, M. 2011. Creation of the WATCH Forcing Data and Its Use toAssess Global and Regional Reference Crop Evaporation over Land during theTwentieth Century. Journal of Hydrometeorology, 12, 823–848. Wescoat, J. L. J., Sarah, J., Halvorson & Daanish, M. 2000. Water Management in the Indus Basin of Pakistan:A Halfcentury Perspective. Water Resources Development, 16, 391–406.
Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

A Gaussian design-storm for Mediterranean convective events

I. Andrés-Doménech, R. García-Bartual, M. Rico Cortés & E. Albentosa Hernández

Universitat Politècnica de València (UPV), Instituto de Ingeniería del Agua y Medio Ambiente (IIAMA),

Valencia, Spain

ABSTRACT: The Spanish Mediterranean façade usually experiences flash floods caused by torrential rain storms. Consequently, achieving a deeper knowledge about rainfall characteristics has been of paramount importance to improve design criteria and to develop more efficient and safer hydraulic infrastructures. This research develops a new framework to define design storms

with temporal and spatial distribution, based on observed rainfall patterns. It is applied to the rainfall features of Valencia, Spain. Convective episodes have been identified from the 1990–2012 period and individual storms have been extracted; statistical description of their internal characteristics (volume, duration and peak intensity) has been performed. Dependencies between the three variables are explored and mathematical relationships proposed. The processed data is finally used to fit the temporal pattern of the theoretical model for the design storm, based on a Gamma function. Results for Valencia data show a good performance of the model, which basically captures the most essential temporal patterns of rainstorms.

1 INTRODUCTION

Design storms were introduced in hydrological engi neering many decades ago to give a practical response to the need of a representative rainfall input for hydraulic infrastructure dimensioning. Obviously, this procedure to obtain design storm needs important sim plifications as it is impossible to represent in a single event such a complex and chaotic process.

Several design storms have been proposed since the last decades (Keifer and Chu, 1957; Marsalek, 1984; Watt and Marsalek, 2013). Usually, design storms usu ally fall into two different categories.The first one con siders models based on intensity-duration-frequency (IDF) relations. The second one corresponds to syn thetic events where the temporal distribution is derived from observed storm patterns.

Within the first category, the alternating block synthetic hyetograph (Chow et al., 1988) has been widely used. The method is simple but also criticised because it does not represent any observed rainfall internal structure. Moreover, the storm duration and consequently the total rainfall depth are arbitrarily selected. In addition, recent publications demonstrate that, generally, peak flow calculations using these design storms tend to overestimate the results (Alfieri et al., 2008).

To overcome these limitations, the second category of design storms aims at reproducing the internal structure of observed episodes. In Spain, García-Bartual and Marco (1990) studied hyetographs of extreme convective precipitation where intensity resulting from the activity of each rainfall cell was represented by a gamma-type function shape with maximum intensity and volume as random variables. This premise is used in this work to propose a design storm for Mediterranean convective events. The Spanish Mediterranean coastline usually experiences flash floods caused by heavy torrential rainstorms. Consequently, achieving a deeper knowledge about rainfall characteristics has been of great interest to improve design criteria and to develop more efficient and safer hydraulic infrastructures. This paper presents a detailed analysis of convective storms in the city of Valencia (Spain), establishing objective criteria to identify these events and analysing its main features and temporal internal patterns. Relationships between the three main descriptors of the storm (rainfall volume, maximum intensity and storm duration) are also explored. Finally, a complete spatial and temporal design storm is proposed and its main properties established. The temporal parameters of the model are fitted according to observed storms within the period 1990–2012, demonstrating the reliability of the proposed design storm model for convective episodes. 2 RAINFALL DATA 2.1 Data set Valencia is located on the eastern Mediterranean coastline of Spain. Its climate is Mediterranean, mild, with some semiarid features (type CSa, temperate with dry and hot summer, according to Köppen-Geiger

classification). Average temperature is around 17 ∘ C

and average annual rainfall depth is close to 450 mm,

with a very strong seasonality. Rainfall storms are usu

ally concentrated in autumn, with typical very high

peak intensities (convective torrential rain).

The rainfall series used in this study was recorded

by the Júcar River Basin Authority during the period

1990–2012. The rainfall gauge is installed in the city

centre and data time step is 5 minutes. Previous studies

demonstrated the validity of this data set for similar

purposes (Andrés-Doménech et al., 2010).

The continuous rainfall series is processed to

identify and extract convective storms. First, statisti

cally independent rainfall events are identified. Then,

amongst them, only convective events are extracted.

Finally, convective storms are identified from con

vective events and finally selected for the model

parameters estimation.

2.2 Selection of independent convective events

Before tackling the storm analysis, separation of the original continuous series of rainfall records in statistically independent rainfall events is a preliminary question. There is not a universal method for the identification of the minimum interevent time of a rainfall regime, for independent storms definition. Works by Restrepo-Posada and Eagleson (1982) are still in force; identification of independent events is based on considering events like statistically independent achievements, so that the minimum interevent time must define a Poisson process. Bonta and Rao (1988) bear out this theory, studying in depth some other aspects. Andrés-Doménech et al., 2010 completed the original methodology based on the coefficient of variation analysis and established for Valencia a minimum interevent time equal to 22 hours. This implies that if two rainfall pulses are separated more than 22 hours, then they belong to different events. Under this premise, 987 statistically independent events are identified for the period 1990–2012.

For the present study, only convective storms are of interest. Llasat (2001) established an objective methodology to assess the convective degree of a rainfall event according to the following index:

where θ (I-L) is a Heaviside function and L a convectivity threshold depending on Δt. For Δt = 5 min,

I = 35 mm/h. $\theta(I-L) = 1$ if $I > L$, $=0$ if $I < L$ and $=1$ for $I = L$. Consequently, this index represents the proportion of total rainfall fallen with an intensity higher than 35 mm/h. This convectivity threshold was estimated for the Spanish Mediterranean coastline by Llasat (2001). Events with β L,▮T ∗ > 0.3 represent convective storms. Thus, according to this additional criterion, only 64 convective events from the complete set are selected. Figure 1. A type I event recorded on 11th October 2007. 2.3 Selection of storms Some of the independent convective events selected above can correspond to long or very long episodes with important dry intra-periods (always lower than 22 hours). Concatenation of some convective cells can lead to this situation, resulting in long episodes of some days. Often, these rainfall cells (storms) can be linked by very slight background intensity (around 2 mm/h). Usually, these convective cells only correspond to a small duration within the whole episode, nevertheless, they can represent more than 80% of the total rainfall amount. According to this fact, the convective events set is classified as it follows: a) Type I events. They correspond to a unique convective cell. Their main feature is a moderate duration and a quite high mean intensity.They can also experience low intensity intervals before and after the peak interval. Figure 1 shows an example of a type I event. b) Type II events. Long rainfall episodes composed of two or more storms as explained before. Figure 2 shows an example of a type II event. Following this classification, 58 events are type I and 6 events are type II.These 6 type II events are carefully examined and analysed to extract storms within them. Criteria to select individual storms are adopted: a) Identify the event peak intensity, always over 35 mm/h and its near range. b) The first storm time interval corresponds to the prior interval to 9.6 mm/h intensity (3 times the rain gauge sensitivity).

Figure 2. A type II event recorded on 1st July 2002.

Table 1. Storm univariate statistics. V S I 0 T S

Variable (mm) (mm/h) (h)

Mean	20.0	76.4	38.0
Maximum	69.2	206.4	115.0
Minimum	4.2	36.0	10.0
Median	15.0	64.8	30.0
Std. Deviation	15.9	37.3	21.9
Bias	1.39	1.46	1.21
Kurtosis	1.36	2.09	1.18

Table 2. Correlation matrix of V_S, I_0 and T_S.

	V_S	I_0	T_S
Variable	(mm)	(mm/h)	(h)
V_S (mm)	1	-	-
I_0 (mm/h)	0.639	1	-
T_S (h)	0.839	0.261	1

c) The last storm time interval is defined by a shift in the sign of the hyetograph derivative, always around intensities lower than 9.6 mm/h.

Finally and according to this methodology, 73 storms are defined for the 1990-2012 period. Table 1 shows statistical descriptors of the main storm features: volume (V_S), maximum (peak) intensity (I_0) and duration (T_S).

Another important previous analysis is the correlation matrix (Table 2). As it can be observed, there is a strong correlation between the storm volume and duration and also an evident link between storm vol ume and its maximum intensity. These correlations are Table 3. Final proposal of intensity thresholds for the relation V S -T S analysis. Threshold Number Range (mm/h) of storms R 2 Low intensity <55 22 0.904 Medium intensity 55-75 22 0.950 High intensity >75 29 0.702 Figure 3. Storm volume-duration relations depending on the storm maximum intensity range. on the basis of the further relations presented in this study. 3 RELATIONS BETWEEN STORM VARIABLES 3.1 Relation volume-duration The high correlation coefficient between V S and T S corresponds to a high linear dependence between these variables. Nevertheless, the scatterplot of the two variables shows a significant dispersion around intermediate volumes. For this reason, and for the sake of a simple model, V S -T S relation is explored dividing the data set according to storm maximum intensity thresholds: low intensity, medium intensity and high intensity. The final selection of these intensity thresholds is summarized in table 3 where the three intensity ranges are defined and the number of storms resulting in each sub-set indicated. Finally, the coefficient of determination R 2 is shown for the linear relation between V S and T S in each sub-set. Obtained R 2 values are higher in each sub-set than in the complete data set. Figure 3 represents the three families and the linear relations V S -T S for each one.

Table 4. Final proposal of duration thresholds for the V S -I 0

analysis. Threshold Number

Range (min) of storms R 2

Short storms <30 33 0.676

Intermediate storms 30-60 26 0.724

Long storms >60 14 0.819

Figure 4. Storm volume-maximum intensity relations

depending on the storm duration range.

3.2 Relation VolumePeak intensity

The correlation matrix (Table 2) also highlights a

high correlation between storm volume and maximum

intensity. Nevertheless, as in the V S -T S analysis, there is an apparent too high dispersion in the complete scat terplot. A parallel analysis is now performed to select duration thresholds to divide the complete set in three sub-sets. Table 4 and figure 4 summarise the results obtained.

The obtained relations, both V_S -T_S for different I_0 ranges or V_S -I_0 for different duration ranges, will be useful in the proposed methodology to define design storms without losing the relations existing between the three main descriptors of the storm.

4 DESIGN STORM MODEL

4.1 Design storm formulation

The theoretical storm model is based on García Bartual (2013). The author proposes a storm model adapted for convective storms with a very well defined

This research presents preliminary results concern ing storm temporal patterns and consequently f(t) and i_0 properties. Further results concerning g(r) esti mation are not addressed herein. Accordingly, there inafter the model is considered at rain gauge scale, so that

As shown above, i(t) is asymptotic to 0 when time increases. Consequently, for practical purposes it has to be truncated. The temporal end of the storm (t_S) is placed when its intensity is 10% of i_0 , so

Finally, the total amount of rainfall v_S (mm) according to the model is

where i_0 is in mm/h and ϕ in min.

4.2 Fitting model parameters

Once the model formulated and its main properties known, parameters i_0 and ϕ are fitted according to the 73 storms available in the data set. Two objective functions have been used: the standard method of least squares (LS-method) and the minimization of a psi function (ψ-method).

The standard method of least squares consists of obtaining the best estimates for i_0 and ϕ that minimize

where $i(j)$ is the empirical intensity for $t = j$ and $i*(j)$ is the model theoretical intensity for the same instant. As it can be observed, this standard procedure only aims at minimizing errors between theoretical and empirical intensities. Nevertheless, other internal features of the storm are of interest to be reproduced by the model.

For this reason a more complex function is proposed to be minimized. The ψ function is defined by:

where $I(j)$ is the empirical intensity for $t = j$, $i(j)$ is the model theoretical intensity for the same instant, V_S is the empirical storm volume, T_S its empirical dura tion, I_0 its maximum intensity and T_P the percentage of the total storm duration till the storm peak; v_S t_S , i 0

and t P are the corresponding theoretical model val

ues. Finally, ω_k (k = 1, …, 5) are non-dimensional

weighting coefficients. According to the importance

of each parameter included in the ψ function, the

following weights are adopted: $\omega_1 = 1/n$; $\omega_2 = 10$; Table 5. Model errors for the LS-method. Absolute errors Relative errors v S i 0 t S v S i 0 t S Statistic (mm) (mm/h) (min) (%) (%) (%) Mean 3.5 32.7 12.7 27.0 53.9 35.0 Max. 36.2 207.8 104.0 165.3 412.3 188.8 Min. 0.0 0.3 0.0 0.0 0.1 0.1 Median 1.6 13.2 10.0 10.8 20.7 32.6 Std. Dev. 5.0 52.0 13.8 38.2 96.0 27.9 Table 6. Model errors for the ψ-method. Absolute errors Relative errors v S i 0 t S v S i 0 t S Statistic (mm) (mm/h) (min) (%) (%) (%) Mean 1.6 6.3 6.5 8.1 7.8 16.2 Max. 7.3 32.1 20.0 37.6 20.6 36.2 Min. 0.0 0.0 0.0 0.6 0.1 0.1 Median 0.9 4.6 5.0 6.3 7.4 16.7 Std. Dev. 1.6 6.4 5.5 6.4 5.0 9.3 $\omega_3 = 2$; $\omega_4 = 5$; $\omega_5 = 1$. Parameters are fitted using both methodologies. Table 5 shows errors achieved using the LS-method whilst table 6 corresponds to errors achieved with the ψ-method. Errors using the LS-method are quite higher for all the storm descriptors. It is especially remarkable that errors for maximum intensities are up to 412% which is unacceptable. This kind of errors corresponds to short duration storms where the theoretical peak occurs between two 5-min intervals of similar intensity. Definitely, the multicriteria ψ-method achieves better results and errors are always lower for each descriptor. Figure 5 and 6 show examples of the model accuracy (ψ-method) for a long and short duration storm respectively. 4.3 Validation of model properties Once the model fitted, expressions given by equations (13) and (14) should be validated. This equations allow to relate the storm parameters to the storm duration and volume. Combined with relations obtained in section 3, they completely define the synthetic storm. Figure 7 depicts the ϕ parameter for each storm duration in the data set. Empirical values are compared to theoretical ones given by equation (13). The validation gives a coefficient of determination $R^2 = 0.896$ and a Nash-Sutcliffe efficiency index NS = 0.832. Both values show a very good accuracy of the model. Figure 8 shows the empirical and theoretical (model) relations between the ϕ model parameter and the storm duration. As results are dependent on

Figure 5. Model fit for the storm on 11th October 2007.

Figure 6. Model fit for the storm on 7th April 2009.

the intensity range considered, each figure represents results for each intensity range considered in table 5. Moreover, the “model upper bound” and “model lower bound” represent the theoretical relation according to equation (14) and for both intensities defining the interval bounds. The graphical analysis highlights again the accuracy of the model. Figure 7. Validation of expression $t_S = 4.89\phi^{-1}$. Figure 8. Storm volume vs. ϕ for the subsets $i_0 > 75$ mm/h (high intensity), $55 < i_0 < 75$ mm/h (medium intensity) and $35 < i_0 < 55$ mm/h (low intensity). Theoretical expression $v_S = 0.0433 \cdot i_0 \phi^{-1}$ computed for the average i_0 value in each subset. 5 CONCLUSIONS The proposed design storm fits accurately the observed temporal pattern of historic convective events. Parameters i_0 and ϕ have been successfully characterised from the data set and their relationships with the other main descriptors of the storm (rainfall volume and duration) established and validated.

Ongoing work is now being developed to com pletely define the practical steps for the proposed methodology: g(r) parameters estimation from radar data and a convenient procedure for the storm return period estimation.

ACKNOWLEDGEMENTS

This study was developed within the project “Formu lación de un hietograma sintético con reproducción de las relaciones de dependencia entre variables de evento y de la estructura interna espacio-temporal”

(GV/2015/064) funded by the government of the

Valencia Region (Conselleria d'Educació, Investi

gació, Cultura i Esport, Generalitat Valenciana).

Alfieri, L., Laio, F., and Claps, P. 2008. A simulation exper

iment for optimal design hyetograph selection. Hydrol.

Process. 22: 813–820. doi:10.1002/hyp.6646.

Andrés-Doménech I., Montanari A. and Marco J.B. 2010.

Stochastic rainfall analysis for storm tank performance

evaluation. Hydrology and Earth System Sciences 14:

1221–1232. Bonta, J. V. and Rao, R. 1988. Factors affecting the identification of independent storm events, J. Hydrol. 98 (3–4): 275–293. Chow, V.T., Maidment, D.R. and Mays L.W.1988. Applied hydrology, Mc Graw-Hill. García-Bartual, R. and Marco, J. 1990. A stochastic model of the internal structure of convective precipitation in time at a raingauge site. Journal of Hydrology. 118: 129–142. Keifer, C.J. and Chu, H.H. 1957. Synthetic storm pattern for drainage design. J. Hydraul Div. Proc. Amer.Soc. Civ. Engrs 83(HY4): 1332-1-1332-25. Llasat M.C. 2001. An objective classification of rainfall events on the basis of their convective features: application to rainfall intensity in the northeast of Spain. International Journal of Climatology. 21: 1385–1400. Marsalek, J. 1984. Temporal distribution of design storm rainfall.Water Science andTechnology. 16(8/9): 167–175. Restrepo-Posada, P. J. and Eagleson, P. S. 1982. Identification of independent rainstorms, J. Hydrol. 55 (1–4): 303–319. Rodríguez-Iturbe, I. and Eagleson, P. S. 1987: Mathematical moels of rainstorm events in space and time. Water Resources Research, 23(1):181–190. Watt, E. and Marsalek, J. 2013. Critical review of the evolution of the design storm event concept. Canadian Journal of Civil Engineering. 40(2):105–113. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Extreme hydrological situations on Danube River – Case study Bezdan

hydrological station (Serbia)

M. Urošev

Geographical Institute “Jovan Cvijic´” SASA, Belgrade, Serbia

I. Lešcˇešen

Department of Geography, tourism and Hotel Management, Faculty of Science, University of Novi Sad, Novi Sad, Serbia

D. Štrbac

Geographical Institute “Jovan Cvijic´” SASA, Belgrade, Serbia

D. Dolinaj

Department of Geography, tourism and Hotel Management, Faculty of Science, University of Novi Sad, Novi Sad, Serbia

ABSTRACT: In this paper we analyzed extreme hydrological situations on Danube River at Bezdan for the 1931–2014 period. Floods and hydrological droughts were defined as events over and below selected threshold levels respectively. The magnitudes (volumes) and durations of floods and droughts were analyzed by annual maximum (AMS) and partial duration series (PDS) methods. In flood frequency analysis of the AMS Log Pearson 3 distribution has the best agreement with empirical data for both volumes and durations, while for hydrological droughts it was the Pearson 3 distribution for Q 90 threshold and Log-Pearson 3 for Q 95 threshold. In flood frequency analyses of the PDS joint distribution of Poisson and Weibull (P +W) fits the empirical distribution of largest volumes and durations, while for hydrological droughts best distribution depends on selected threshold levels. Comparative view of calculated

return periods by AMS and PDS methods for both floods and hydrological droughts is presented.

1 INTRODUCTION

People around the world are exposed to various nat ural hazards, such as earthquakes, volcanic erup tions, hurricanes, storms, tornadoes, floods, and droughts. Hydrological extremes (floods and hydro logical droughts) are natural hazards that are not limited to specific regions, but occur worldwide and therefore, impact a very large number of people. Floods receive most attention, both in the news and in scientific literature, due to their fast, clearly visible, and dramatic consequences. Droughts develop slower, often unnoticed and have diverse and indirect conse quences (Van Loon, 2015). In the Danube River basin and in the basins of the other major European rivers, extreme hydrological events happened over the latest years, are evidence of new tendencies in the meteo rological and hydrological processes. Global climate change, intensification of synoptic processes, change in the total amount of precipitation and its irregularity resulted in an increase in a frequency of extreme hydro logical events (Schneider et al. 2011, Doll & Zhang 2010, Nohara et al. 2006). There are many articles that focused on changes in flood hazard (Kundzewicz 2015, Kundzewicz & Matszak 2015, Gyawali et al. 2014, Vogel et al. 2011, Dankers & Feyen 2009) and hydrological droughts hazards (Prudhomme et al. 2014,Van Huijgevoort et al. 2014, Orlowsky & Seneviratne 2013, Wanders et al. 2014, Feyen & Dankers 2009). Floods and hydrological droughts affect many aspects of environment and society, and future increase in the demand for water will be most critical in periods of these extreme hydrological events. Information on characteristics of these extremes provides critical values for different water management strategies. In terms of hydrological research there is a need to improve our ability to predict the onset, duration and severity of a flood or drought, thus providing a better basis for the design of water management facilities. Frequency analysis of historical data provide an important contribution to improve our knowledge of the status and dynamics of water resources. First step in analyses of floods and droughts is to define them. Floods and drought are complex phenomenon and they could be defined in many ways. Although generating processes of floods and hydrological droughts are different, in this study, in order to be persistent, we defined them in similar way.

Figure 1. Mean monthly discharges of Danube River at

Bezdan.

Floods are defined when water discharge is above predefined threshold level, and drought when the dis charge is lower than threshold. They are defined as two random variables X = (V, T), where V = volume of flood or drought; T = duration of flood or drought. This definition of floods and drought is more use ful to hydrological engineers than a single value of instantaneous water discharge. There are much less scientific articles with frequency analysis of volumes and durations of floods and droughts than ones deal ing only with water discharges. That’s why we present both annual maximum series (AMS) and partial dura tion series (PDS) methods for modeling at-site flood and drought volumes and durations on the example of Bezdan hydrological station in Serbia.

2 DATA

Bezdan gauging station is the oldest hydrological sta tion on Serbian part of Danube River, established in 1856. It is located on 1425.6 km from river mouth, just as Danube enters Serbia, with drainage area of 210,250 km^2 and lies at 80.64 m above sea level (www.hidmet.gov.rs). In the period 1961–2014 mean annual water discharge is 2296 m^3/s, coefficient of variation (C_v) 0.19, and it has a positive, but not statistically significant, trend of mean annual dis charges. Annual maximum discharge with exceedance probability 1% ($Q_{max1\%}$) is 8803 m^3/s, while annual minimum discharge with probability 95% ($Q_{min95\%}$) is 788 m^3/s (Urošev et al., in press).

Seasonal variations in discharges depend on pluvial and air temperature regime in basin, upstream of gaug ing station. In case of Bezdan, it depends on climate regime of large area upstream, where Alps are domi nant factor. High waters occur in period between April and June, with maximum in June, while low water period is characteristic for October and November, with minimum in October (Fig. 1). That’s why cal culations in this paper are done by calendar years (01.

January–31 December). Bezdan gauging station has

good quality long series of discharges. The 1931-2014

period was chosen because for the frequency analy

sis of flood and drought volumes and durations it is necessary to have good quality daily data series as long as possible. In Serbia, as in other countries, more attention was given to frequency analysis of AMS and PDS of water discharges, while there are only few of them related to the calculation of distribution functions of volumes and durations of extreme hydrological events (Pavlović et al. 2014, Zelenhasić 2002, Salvai et al. 1990, Zelenhasić & Salvai 1987). 3 METHODS 3.1 Threshold method for definition of floods and droughts Method of “threshold level” was used mostly for definition of hydrological droughts, mainly because its results have large number of application in low water domain rather than for floods. Volumes (deficits) and durations of droughts are important information for reservoir management, drinking water supply, irrigation, navigation, water quality management and hydropower. Here we will present this method for definition of drought, wile for definition of floods this method could be also used by applying only different threshold level. Method of threshold level (also called “truncation level”) is explained in detail in the works of Tallaksen et al (1997) and Zelenhasić & Salvai (1987). Intensive use of this method starts after year 1987, i.e. after the publication of Zelenhasić & Salvai (1987), who first applied the threshold method on daily data. Later modifications of the threshold method relate mainly to the way in which mutually dependent droughts are merged and minor deficits are eliminated, and also to the choice of a theoretical distribution function of maximum deficit and duration of droughts. Manual of the low-flow estimation and prediction of the World Meteorological Organization (WMO, 2008) also recommends a threshold method for selection of hydrological droughts. The method comprises of applying predefined threshold level on the hydrograph, so all discharges that are below the threshold are considered droughts. Selection of a threshold level is crucial for the use of threshold level method. Ideally the threshold level should be related to some drought impact, e.g. irrigation water requirements, drinking water supply, reservoir operation levels, minimum water depth for navigation, or environmental flows to support stream ecology (Van Loon, 2015). Often, this information is not available or the drought analysis aims at a number of users with different requirements and, therefore, different threshold levels. Consequently, for practical reasons thresholds are often derived from percentiles of the flow duration curve (Fig. 2), commonly ranging between the 70th and 95th percentile for perennial rivers. According to Fleig et al. (2006) there is no single threshold level that is preferable and the selection of a specific threshold level remains a subjective decision. In the

Figure 2. Flow duration curve of Danube River at Bezdan.

analyses of extreme droughts the most frequent threshold used is Q_{95}, i.e. discharge that is equal or exceeded 95% of observation period. Therefore, in this paper we used Q_{95} and Q_{90}, from the flow duration curve of Danube River at Bezdan for period 1931–2014, for the threshold level for selecting hydrological droughts and Q_5 and Q_{10} for selecting floods (Fig. 3). We chose this very high and very low threshold because our goal is to model extreme floods and droughts, which have moderate and large return periods and also to see how different thresholds influence extreme value modelling.

The use of a daily time resolution introduces two special problems: dependency among droughts and the presence of minor droughts. During a prolonged dry period it is often observed that the flow exceeds the threshold level for a short period of time and thereby a large drought is divided into a number of minor droughts that are mutually dependent. To avoid these problems that could distort an extreme value modelling, some kind of pooling procedure needs to be done in order to define an independent sequence of droughts (Tallaksen, 2000).

Tallaksen et al. (1997) compared and described three different pooling procedures: the moving average procedure (MA), the sequent peak algorithm (SPA) and the inter-event time and volume criterion (IC). The use of MA and SPA both proved satisfactory with respect to pooling dependent droughts and reducing the number of minor droughts. The MA procedure is applied to the time series prior to selecting the droughts. In this case the time series is smoothed and minor peaks above the threshold level removed. It is recommended to use moving average interval of 10 days (Hisdal & Tallaksen, 2000). In this paper we used a central moving average with interval of 11 days (MA (11)), so that actual dates of selected droughts and floods are not shifted. Now we can easily determine drought and flood characteristics: volume (deficit and surplus, respectively) and duration. Although MA(11) filter removes great number of minor droughts and floods, and merged dependent events, number of those events still remains. That’s why we need additional criterions for dealing with them. The first criterion is inter event time criterion

(t_c) > 5 days, because of the MA (11) filter applied to the series. Fleig et al. (2006) analyzed drought in basin all over the word and concluded that for excluding minor droughts the most suitable combination is minimum duration of drought (t_{min}) > 2 days and minimum deficit (D_0) = $0.005 \times D_{max}$, where D_{max} is maximum observed deficit. For the Danube River at Bezdan we also used these two criterions for removing minor droughts. All three criterions were also applied for floods. These last two criterions t_{min} and D_0 also serve as location parameter for the distribution functions of exceedance magnitudes of duration and volume in the PDS model. 3.2 Extreme value modeling The two most applied models for extreme value analysis are the annual maximum/minimum series (AMS) model, and the partial duration series (PDS) model. In the AMS model the largest/smallest event within a year is selected, whereas the PDS considers all events below/above a given threshold. The PDS model ensures that the most severe events are selected for the extreme value analysis, independent of the time of occurrence, and hence intuitively provides a more consistent definition of the extreme value region than the AMS model. Minimum flows in very wet years might not belong to the low extreme population, as maximum flows in very dry years might not belong to the high extreme population. A drawback of the PDS model is that independent events have to be identified, thus making the selection of events more complicated than in the AMS (Tallaksen, 2000). In case of choosing low threshold levels for the definition of droughts, large number of zero-drought years may significantly reduce the information content of the AMS, and vice versa for floods, high threshold levels lead to the occurrence of zero-flood years. In the PDS, however, minor droughts and floods may significantly distort the extreme value modeling, and a procedure for exclusion of minor droughts and floods should be imposed. Whether to use an AMS or PDS model depends on the available data and the type of analysis to be carried out. Frequency analysis of hydrological data requires that the data are homogeneous (from the same population), independent (no serial correlation in the data) and stationary (no statistically significant trend). For testing data on independence we used the first-order serial correlation test (Anderson’s test) and the test of square of consecutive differences (Neumann’s test). For testing data on homogeneity we used: Fischer’s f-test for testing variance, Z-test and Student’s t-test for testing mean value, Mann–Whitney–Wilcoxon Utest test for testing distribution function. For testing data on stationarity we used coefficient of correlation with t-test. 3.2.1 AMS Annual maximum/minimum series (AMS) model is most frequently used in flood and low water frequency

Figure 3. Definition of floods and hydrological droughts on Danube River at Bezdan for the 1931–2014 period.

analysis, mainly because AMS can easily be obtained from the hydrologic time series, as long as the water year is properly defined. Drawbacks of AMS model are presented earlier. There are many ways to select which distribution are gone be used in frequency anal yses of AMS. In some countries, they are defined by official guidelines like USWRC (1982), which sug gests using Log-Pearson type 3 (LP3) distribution for flood frequency analysis (FFA) in USA, or General ized Extreme Value (GEV) and generalized logistic (GL) distributions were proposed in the Flood Esti mation Handbook (Robson & Reed, 1999) for FFA on data from the rivers in Great Britain, or Gosstro Possii (2004) which advocates to use the three parameter generalized gamma (GG3) (also known as Kritsky-Menkel distribution) for FFA in Russia. In Serbia still there is no official guideline for FFA or low water frequency analysis, but there is an unwrit ten rule to use five common distribution N(Normal), LN (Log-Normal), G (Gumbel), P3 (Pearson type 3) and LP3. Those five distributions were used for AMS modeling of flood volumes and durations, which were determined earlier from threshold levels Q_5 and Q_{10} and drought volumes and durations, which were determined earlier from threshold levels Q_{95} and Q_{90} (see Section 3.1). Formulas of used cumulative distribution function (CDF) and their quantile estimator are not presented here, but can easily be found in literature, for example in Stedinger et al. (1993).

3.2.2 PDS

As an alternative to the AMS approach in hydrologic frequency modeling, the partial duration series (PDS) method, also denoted the peaks over threshold (POT) method, has been recommended. The application of method in FFA is discussed in details in Lang et al. (1999). In Section 3.2 we mentioned advantages and drawbacks of PDS model. There are not so many papers that use PDS to model floods and droughts, defined as volumes and durations, in comparison with ones, defining them as single discharge values. In this paper we used criterions described in Section 3.1 to overcome issues related to independence criteria and threshold selection. Inter event time criterion (t_c) > 5 days, was used to select independent floods and droughts. Minimum duration of drought and flood (t_{min}) > 2 days and minimum volume of drought and flood (V_0) = $0.005 \times V_{max}$ were used to select thresholds for series of duration and volumes, respectively. In PDS modeling, instead of threshold, we will use term “base level” for t_{min} and V_0, to make a distinction from threshold levels (Q_{95}, Q_{90}, Q_{10}, and Q_5) used for definition of droughts and floods. The choice of the distribution function of drought and flood volumes and durations can be based on theoretical considerations or on results of goodnessof-fit tests. Theoretical considerations are present in the works of Zelenhasić (Zelenhasić 2002, Salvai et al. 1990, Zelenhasić & Salvai 1987), which uses the theory of random number of random variables (Todorović, 1970). He concluded that the number of occurrences of drought episodes has a Poisson distribution

(P), duration and volume deficits have exponential (E), and the largest deficits and the longest duration of a drought for a certain period of time (usually a year) (annual maxima) have a double exponential distribution (Gumbel (EV1)). The same distributions were used in PDS modeling of floods (Todorović & Zelenhasić, 1970). Many papers have focused on the Generalized Pareto (GP) distribution

Table 1. Characteristic of AMS and PDS samples of flood and drought volumes (V) ($\times 10^9$ m^3) and durations (T) (days) for different thresholds.

	Q 5				Q 10				Q 90				Q 95			
	AMS		PDS		AMS		PDS		AMS		PDS		AMS		PDS	
Characteristic	V	T	V	T	V	T	V	T	V	T	V	T	V	T	V	T
Sample size	42	42	64	64	54	54	94	94	55	55	92	92	30	30	53	53
Mean	1.50	24.1	1.09	20.3	2.57	35.3	1.77	27.9	0.63	36.3	0.48	29.3	0.41	30.6	0.30	24.5
STDEV	2.29	18.2	1.94	16.0	3.23	29.2	2.64	24.2	0.71	27.9	0.62	24.5	0.45	17.9	0.38	16.0
Skewness	4.59	2.81	5.42	3.23	4.03	1.88	4.88	2.58	1.85	1.32	2.28	1.72	2.27	1.05	2.74	1.47
Kurtosis	25.4	11.4	36.0	15.2	21.8	3.88	32.9	7.96	4.08	1.57	6.30	3.29	5.69	1.54	8.77	3.02

for exceedance magnitudes (Madsen et al. 1997, Wang 1991, Hosking & Wallis, 1987). The PDS model of P distribution for number of occurrences with GP distribution of exceedance leads to a GEV distribution of annual maxima floods. For PDS modeling of flood and drought volumes and durations of Danube River at Bezdan we used Poisson and Binomial (B)/ Negative Binomial (NB) distribution for number of occurrences; Exponential, Weibull (W) and Generalized Pareto distributions for exceedance magnitudes;

and for the annual maximum all possible combina tions of selected distributions. Goodness-of-fit tests were used to choose one distribution which has the best fit to the empirical data.

3.2.3 Parameter estimation

There are three main methods for estimating param eters of distribution: method of moments, method of L-moments and method of maximum likelihood.

The main advantage of L-moments over conventional moments is that L-moments, being linear functions of the data, suffer less from the effects of sampling variability. They are more robust than conventional moments to outliers in the data and enable more secure conclusions to be made from small samples about an underlying probability distribution. L-moments some times give more efficient parameter estimates than the maximum likelihood estimates (Hosking, 1990). In this paper L-moments were used for parameter esti mation in both AMS and PDS model. As for CDF, due to limited length of paper formulas for parameter esti mation via L-moments for distributions used in this paper are not presented here, but can easily be found in literature, for example in Hosking & Wallis (1993).

3.2.4 Goodness-of-fit tests

Main purposes of this article is to find which distribu tion functions give the best fit with the AMS and PDS samples for different threshold levels. That's why we used chi-squared (χ^2) test for discrete distributions, for testing distributions of number of drought and flood occurrences in PDS, and Kolmogorov-Smirnov (K-S) and Cramer-von Mises (CvM) for continuous distributions in both AMS and PDS. 4 RESULTS For selected thresholds (Q_{95}, Q_{90}, Q_{10}, and Q_5) eight series were gained all with two variables volume and duration. Overall frequency analysis of 16 series was done. Main characteristics of AMS and PDS samples are shown in Table 1. According to tests, besides F test, all 16 series are homogeneous, independent and stationary. F test gave negative results (variances are unequal) for variances, because hydrological data series often do not satisfy some conditions for strict applicability of parametric test, particularly the assumption of normality. Because the analyzed samples are homogenous and stationary, frequency analysis can be performed. The K-S and CvM tests were applied to the results of flood and drought frequency analysis of AMS series. Critical values are calculated for the 0.05 significance level.According to these tests and probability plot best fit to the empirical data has LP3 distribution (5 times), than P3 (2 times) and LN (ones). They are compared to results of PDS model. According to the results of χ^2 tests number of occurrences of floods and droughts has a P distribution (4 times), than NB (3 times) and B (ones). The K-S and CvM tests and probability plots singled out Weibull distribution for modeling exceedance magnitudes (peaks). For the largest exceedance (annual maxima) combination of Poisson and Weibull distribution (P +W) has best fit to empirical data 4 times, Negative Binomial and Weibull (NB +W) 3 times and Binomial and Exponential (B + E) once. These combinations of distributions are compared to AMS. 4.1 Comparison between AMS and PDS Similar results, according to values of K-S and CvM tests (Table 2) and probability plots (Fig. 4), were obtained for both AMS and PDS model, except for flood durations (Fig. 5). They are showing that just a little bit better fit to empirical data have distributions from AMS model. We should bear in mind that this goodness-of-fit tests are related to whole range of observed values, while in tails of distributions, there are much bigger differences, especially in the upper

Table 2. Goodness-of-fit test results for AMS and PDS model. value

Variable	Threshold	Test	AMS	PDS	Critical
Volume	Q 5	K-S	0.041	0.047	0.210
Volume	Q 5	CvM	0.012	0.016	0.462
Duration	Q 5	K-S	0.036	0.048	0.210
Duration	Q 5	CvM	0.010	0.017	0.462
Volume	Q 10	K-S	0.042	0.044	0.185
Volume	Q 10	CvM	0.021	0.019	0.462
Duration	Q 10	K-S	0.036	0.075	0.185
Duration	Q 10	CvM	0.016	0.098	0.462
Volume	Q 90	K-S	0.061	0.062	0.183
Volume	Q 90	CvM	0.041	0.055	0.462
Duration	Q 90	K-S	0.051	0.048	0.183
Duration	Q 90	CvM	0.028	0.028	0.462
Volume	Q 95	K-S	0.023	0.029	0.248
Volume	Q 95	CvM	0.008	0.010	0.462
Duration	Q 95	K-S	0.045	0.051	0.248
Duration	Q 95	CvM	0.017	0.020	0.462

tail, i.e. in the values of small exceedance probabilities (larger return periods). For all analyzed series differ ences between estimated values for AMS and PDS series are increasing with larger return periods, and in general they also increase with lowering threshold level for flood definition, and increasing threshold for drought definition.

The estimated values of flood volumes, for the threshold Q_5 (4330 m^3/s), are slightly higher in PDS model for smaller return periods (V_{10} and V_{20}), while AMS model gave slightly higher values for larger return periods (V_{100}, V_{200}, V_{500}). Differences vary from 0.2% to 18.2%. Almost the same conclusions could be made for threshold Q_{10} (3770 m^3/s), only that for V_5 and V_{10} values are almost equal, while for other return periods AMS model has slightly higher values. Also the differences for same return periods are slightly higher than ones for threshold Q_5. As we can see from Figure 4 maximum flood volume of 14,403 × 10^6 m^3 was in year 1965. This, high value is not outlier according to Grubbs and Beck test, but it causes difficulties to fit a distribution to the sample. In AMS model (LP3), with floods defined by threshold Q_5, this flood volume has a return period of 691 year, and for threshold Q_{10} the return period is 677. For PDS model (P+W) return periods were even larger, 1037 and 1334 years respectively. From the frequency analysis of flood volumes we can see that 1965 year was both largest in series generated from threshold Q_5 and Q_{10}, while for instance 2006 and 2013 year, sec ond and third largest flood volumes from Q_5 series are only fourth and eighth largest in Q_{10} series. In 2013 flood maximum instantaneous water discharge was 8410 m^3/s, greater than 8360 m^3/s which was recorded in 1965, had a return period of 81 years. If we com pare that to return period of its flood volume: 17 years from Q_5 series and 10 years from Q_{10} series we can see that it was not such a large flood as analysis of

Figure 4. Comparison between AMS and PDS model for flood volume frequency analysis for thresholds Q_5 and Q_{10}.

Figure 5. Comparison between AMS and PDS model for flood duration frequency analysis for thresholds Q_5 and Q_{10}.

maximum instantaneous water discharge points out. Also the analysis of duration of 2013 flood confirms that, it was only the 16th (Q_5) and 22th largest (Q_{10}). This flood (2013) could be an example of importance of flood volume and duration frequency analysis for water resources engineers and decision makers. AMS provides better estimates of return periods of flood durations than PDS model for both Q_5 and Q_{10} thresholds, as it can be seen on Figure 5. Differences between estimates of flood durations from PDS and AMS models are much bigger than for flood volumes (from 2.5% to 45.5% in Q_5 and from 0.0%–74.1% in Q_{10} series). Frequency analyses of flood duration for both thresholds also singled out 1965 flood and in addition flood from year 1944 for Q_{10} threshold. The estimated values of drought volumes and durations, for both threshold Q_{90} (1260 m^3/s) and Q_{95} (1100 m^3/s), are slightly higher in PDS than is AMS model for almost every return period. As opposite to floods differences between estimates of drought volumes from PDS and AMS models are much bigger

than for drought durations. For the threshold Q_{95} the most severe drought was in 1947 with deficit volume of 1930×10^6 m^3 (return period 316 years) and dura tion of 81 day (return period 394 years), followed by 1954 drought with deficit volume of 1740×10^6 m^3

(return period 188 years) and duration of 72 day (return period 160 years). These two droughts change places in frequency analysis of drought volumes for thresh old Q 90 series, with return periods of 123 years for 1954 drought, and 118 for 1947 drought. Drought duration of 1954 is also singled out in Q 90 series. The volumes and durations of drought are in general more correlated, then volumes and durations of floods, which is the result of differences in temporal and spa tial processes that generate hydrological drought and floods.

5 CONCLUSIONS

This paper presents advantages of flood and drought frequency analysis defined by two variables: volume and duration. Usually these two hydrological extremes are analyzed separately, but we tried in this paper to define them in similar way, which allows us to com pare the results of frequency analysis. AMS and PDS methods were compared in order to conclude which gives better approximation to the empirical distribu tion. We presented this method for at-site modeling on the example of Danube River at Bezdan.

In AMS model best fit to the sample data has LP3 distribution (5 times), than P3 (2 times) and LN (ones). In PDS model Weibull distribution gave good results for modeling exceedance magnitudes (peaks).

For the largest exceedance (annual maxima) combi nation of Poisson and Weibull distribution (P +W) has best fit to empirical data 4 times, Negative Bino mial and Weibull (NB +W) 3 times and Binomial and Exponential (B + E).

Overall we can say that based on calculations for two thresholds both AMS and PDS models give sat isfactory estimates of return periods of flood and drought volumes for almost whole range of return peri ods, except for the larger ones like V_{100}, V_{200}, V_{500}, and V_{1000} where AMS model performs better. AMS provides better estimates of return periods of flood durations than PDS model for both Q_5 and Q_{10} thresh olds, while it’s opposite for drought durations – PDS generates more accurate estimates of return periods than AMS model for both Q_{90} and Q_{95}.

Choice of distribution in both AMS and PDS model depends on the data sample. When at-site data sam ple is not adequate or sufficient it’s recommended to use regional data if available. That’s why more and more at-site frequency analysis needs to be done to make conclusions about regional distributions of hydrological extremes.

The frequency analysis pointed out the most

extreme hydrological events on Danube River at Bez

dan for 1931–2014 period – the 1965 flood, being the

Hosking, J.R.M. &. Wallis, J.R 1987. Parameter and quan

tile estimation for the generalized Pareto distribution.

Technometrics, 29(3): 339–349.

Hosking, J.R.M. & Wallis, J.R. 1997. Regional frequency

analysis: an approach based on L-moments. Cambridge

University Press.

Kundzewicz, Z.W. 2015. Climate change track in river floods

in Europe. Proceedings of IAHS, 369: 189–194.

Kundzewicz, Z.W. & Matczak, P. 2015. Extreme hydrological

events and security. Proceedings of IAHS, 369: 181–187.

Madsen, H., Rasmussen, P., & Rosbjerg, D. 1997. Compari

son of annual maximum series and partial duration series

methods for modeling extreme hydrological events. 1. At

site modeling.Water Resources Research, 33(4): 747–757.

Nohara, D., Kitoh, A., Hosaka, M., & Oki, T. (2006).

Impact of climate change on river discharge projected by

multimodel. Journal of Hydrometeorology, 7: 1076–1089.

Orlowsky, B. & Seneviratne S.I. 2013. Elusive drought:

uncertainty in observed trends and shortand long

term CMIP5 projections. Hydrology and Earth System

Sciences, 17: 1765–1781.

Pavlovic´, D., Vukmirovic´, V., Plavšic´, J. & Despotovic´, J.

2014. Method of peaks – one stochastic model for vol ume exceedances (in Serbian). Journal of Faculty of Civil Engineering, 24: 85–91.

Prudhomme, C., Giuntoli, I., Robinson, E.L., Clark, D.B., Arnell, N.W., Dankers, R,. Fekete, B.M., Franssen, W., Gerten, D., Gosling, S.N. et al. 2014. Hydrological droughts in the 21st century, hotspots and uncertainties from a global multimodel ensemble experiment. Proceed ings of National Academy of Sciences of the USA, 111: 3262–3267.

Robson, A.J. & Reed, D.W., 1999. Statistical procedures for flood frequency estimation. Volume 3 of the Flood Esti mation Handbook. Wallingford: Centre for Ecology & Hydrology.

Salvai A., Srd¯evic´ B., & Zelenhasic´ E. (1990). Low flows of larger Pannonia rivers (in Serbian). Vodoprivreda, 22 (125–126): 403–417.

Schneider, C., Fl¨ orke, M., Geerling, G., Duel, H., Grygoruk, M., & Okruszko, T. 2011: The future of European flood plain wetlands under a changing climate. Journal of Water and Climate Change, 2: 106–122.

Stedinger, J.R., Vogel, R.M. & Foufoula-Georgiou, E. 1993. Frequency analysis of extreme events. Chapter 18 (1–66). In D.R. Maidment (ed.), Handbook of Applied Hydrology. New York: McGraw-Hill.

Tallaksen, L.M. 2000. Streamflow drought frequency anal

ysis. In J.V. Vogt & F. Somma (eds), Drought and Drought Mitigation in Europe. Netherlands: Springer Science + Business Media Dordrecht. Tallaksen, L.M., Madsen, H. & Clausen, B. 1997. On the definition and modelling of streamflow drought duration and deficit volume. Hydrological Science Journal, 42(1): 15–33. Todorovic´, P. 1970. On some problems involving random number of random variables. Annals of Mathematical Statistics, 41(3): 1059–1063. Todorovic´, P. & E. Zelenhasic´, E. 1970. A stochastic model for flood analysis. Water Resources Research, 6(6): 1641– 1648, 1970. Urošev, M., Kovacˇevic´-Majkic´, J., Štrbac, D., Milanovic´ Peši c´, A., Milijaševic´, D., Jakovljevic´, D. & Petrovic´, A. In press. Waters of Serbia (in Serbian). In M.Radovanovic´ (ed.), Geography of Serbia. Belgrade: Geographical institute "Jovan Cvijic´" SASA. USWRC (United States. Interagency Advisory Committee on Water Data. Hydrology Subcommittee). 1982. Guidelines for determining flood flow frequency. Reston, VA: US Department of the Interior, Geological Survey, Office of Water Data Coordination. Van Huijgevoort, M.H.J., Van Lanen, H.A.J., Teuling, A.J., Uijlenhoet, R. 2014. Identification of changes in hydrological drought characteristics from a multi-GCM driven ensemble constrained by observed discharge. Journal of Hydrology. 512: 421–434. Van Loon,A.F. 2015. Hydrological drought explained.WIREs Water, 2: 359–392. Vogel, R.M., Chad, Y. & Meghan, W. 2011. Nonstationarity: Flood Magnification and Recurrence Reduction Factors in the United States. Journal of theAmericanWater Resources Association (JAWRA), 47(3): 464–474. Wanders, N., Wada, Y., Van Lanen, H.A.J. 2014. Global hydrological droughts in the 21st century under a changing hydrological regime. Earth System Dynamics Discussion, 5: 649–681. Wang, Q.J. 1991. The POT model described by the generalized Pareto distribution with Poisson arrival rate. Journal of Hydrology, 129: 263–280. WMO 2008. Manual on low-flow estimation and prediction. Operational hydrology report No. 5. WMO –No. 1029. Geneva, Switzerland. Zelenhasic´, E. & Salvai, A. 1987. A method of streamflow drought analysis. Water Resources Research, 23(1): 156– 168. Zelenhasic´, E. 2002. On the extreme streamflow drought analysis. Water Resources Management, 16: 105–132. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Presenting an empirical model for determining the sugar beet

evapotranspiration by GDD parameter (Case study:

Torbat-Jam, Iran)

F. Jan Nesar, A. Khashei Suiki, S.R. Hashemi & S. Moradi Kashkooli

Department of Water Engineering, Faculty of Agriculture, University of Birjand, Iran

ABSTRACT: So far, various methods were presented for determining the reference crop evapotranspiration in

different parts of the world.The most popular and prestigious of them can be combined methods of Penman family,

modified Blany-Criddle, Hargreaves – Samani and thornthwaite. Due to the lack of the lysimeter data in many

parts of Iran, presenting an equation according to the regional condition with high precision is so paramount. The

aim of this paper is presenting a method with higher precision for determining the evapotranspiration of the sugar

beet inTorbat_Jam by using the meteorology and GDD(Growing Degree Day) parameter and determined methods

were evaluated with FAOPenmanMonteith method as a standard model for determining the evapotranspiration.

RMSE, R^2 indexes, and Nash-Sutcliffe index, NS, were used for comparing fitness indexes. Results showed that

number four equation is appropriate for calculating the water requirement of crop. So R^2, RMSE indexes and

Nash-Sutcliffe coefficient were 0.683, 1.117 and 0.99, respectively for equation 9.

1 INTRODUCTION

Evapotranspiration is one the most important parame

ters for determining of water requirement crops and

designing the irrigation systems. Determining the

amount of water that is used for evapotranspiration accurately, is one of the most paramount elements for planning and achieving more yield and due to the lack of water resources for agriculture in Iran, accu rate determining of water requirement crops is very useful for saving the water usage. The amount of the evapotranspiration can be determined by implying the specific crop coefficient and potential evapotranspira tion. Regarding to the water critics that is an important problem in the next decades, planning with higher pre cision for optimum usage of available water resources is very significant especially in agriculture section that consists the great amount of water consumption. In order to achieve this aim, the first step is determin ing the amount of water requirement crops and garden plants (Mohseni et al, 2013).

Sugar beet is of the most important sources for pro ducing sugar in the world (Rahimian et al, 2000). So investigation and planning to increase the production of this product for Self-sufficiency the country and preventing the import of sugar is very essential. Nec essary studies for this approach are determining the water requirement for this crop. However the main aim of the irrigation is supplying the water require ment of the plant (evapotranspiration), the importance of determining the water requirement crop is not cared sufficiently in irrigation plans that causes the adverse

proportion between the cultivated land and water in other words the project is designed more or lower than adequate amount that both of these, wastes the investments. If a little amount of cost is allocated for determining plants water requirement before the implementation and in designing step, the mentioned problems will not happen. Reasons of the existence of errors in determining plant water demands are lack of adequate knowledge of designers in agronomical problems and plant physiological properties in relation to the water consumption and selecting the bad method for determining plants water demand (Shahabi far et al 2004). Sugar beet water requirement is mainly depends on the meteorological condition, irrigation management and the length of the growth period also density and the amount of the consumed nitrogen (Koocheki, et al, 1993). Due to the long growth period, sugar beet is counted as a plant with high consumption so that the amount of its water requirement was measured 883 and 762.8 mm in the growth period in Karaj and Mashhad, respectively (Ghalebi 2000, Rahimian et al, 2000). Reference crop evapotranspiration is defined as the amount of the evapotranspiration from the surface that completely covered with the grass with 8 to 15 cm height when there is no water crisis. Evapotranspiration is a zonal not local phenomenon and according to the local changes of the effective factors in evaporation such as evaporated surface and effective environmental factors in measuring the amount of the evaporation or evapotranspiration, distributional equation should be done (Allen, et al, 2002). In order to calculate the amount of evapotranspiration of various plants such as maize and sugar beet, some models with

meteorological parameters basis can be used (Jensen

et al, 1990). These models measure the amount of the

evapotranspiration and if this amount is multiplied to

the crop coefficient, the amount of the plant evapo

transpiration or ETc can be calculated (write et al,

1982).Approximating Kc is a complex process accord

ing to the actual evapotranspiration for the crops that

is done by the management of the aquatic communi ties (Air mark et al, 2013). Growth and development of the crops is usually depends on the time opera tion. Although this approach can be inadequate, this is because of the temperature that varies from one year to the other. Due to the global changes of the climate, usage of the Growing degree days (GDD) concept is expanded that consists of the time and temperature information that plant experiences in the length of the growing period. The needed water for one product is calculated by multiplying the reference evapotranspi ration and crop coefficient and this has a paramount role in the hydrological rotation management in farm ing lands (Sou et al, 2013). Estimating the amount of the transpiration is essential for the water level, agri cultural water, water resources, flow forecasting stud ies and planning for the groundwater sources and this plays important role in agriculture and water resources management (Samaei et al, 2013).Availability of many equations for estimating evapotranspiration, requires abundant information and adequate experience for cor rect usage of the various equations and this is difficult to select the most appropriate method among the men tioned methods. Methods of the evapotranspiration estimation is studied by many researchers (Morton et al, 1994). Sentnelhas et al (2010) carried out a study in order to check the efficiency the evapotran spiration estimation by four methods: FAOPenman Monteith, Priestley-Taylor, Hargreaves, and Thornth waite. Results showed that if there are related data to the wind speed, net radiation and vapor pressure deficit, FAOPenman Monteith has the best efficiency such that the amount of RMSE is 0.53 mm per day and if there are related data to the wind speed and vapor pressure deficit, or if there are the data related to the temperature, Priestley-Taylor and modified Thornth waite has the best efficiency with the amounts of RMSE 0.4 and 0.74 mm per day, respectively. One way to achieve the stable agriculture is evapotranspi ration management that this management can be done in zonal scale and in acceptable frame (Kaviani et al, 2013). Most of studies suggested combinational meth ods especially PenmanMonteith method in expanded domain of climates (Gavilan et al, 2007). According to the studies that have been carried out and water cri sis and the necessity of exact estimation of the ET0 amount, the aim of this study is presenting the new equation that sugar beet water requirement can be esti mated with more precision in Torbate-Jam zone (Iran) so the GDD parameter was used in order to determine equations with more precision as the daily amount of the sugar beet evapotranspiration can be determined

by equations that in which GDD parameter is used and

results will be compared with the evapotranspiration Figure 1. Geographical location of Study area. amount determined by FAO-PenmanMonteith and daily data were used for evapotranspiration. 2 MATERIALS AND METHODS This study has been carried out in Torbate-Jam synoptic stations (35 ∘ 15 ′ N, 60 ∘ 35 ′ E) and 950.4 meters height (Fig. 1). In this study, FAO-Penman-Monteith was used as a standard method due to the absence of the lysimeter data and presence of the complete meteorological data of synoptic station and according to the FAO suggestion, the accuracy of all determined equation were compared with this equation. The length of statistical period was 18 years (1993– 2011) and all of these equations were defined by these data. Therefore in order to determine the evapotranspiration, 21 effective climatic factors were used that consisted of minimum temperature, maximum temperature, average temperature, the sunny hours, minimum humidity, maximum humidity, average humidity, wind speed, latent evaporation heat, slope of the saturation vapor pressure temperature relationship, saturated vapor pressure, actual vapor pressure, vapor pressure deficit, Solar radiation angle, the relative distance between sun and earth, the hourly angle of sunset, extraterrestrial radiation, the hours of day light, net radiation and soil heat flux. 2.1 Determining the regional equations for calculating evapotranspiration At first, In this study, climate factors were collected from synoptic station of Torbat-Jam in a 18 years period (1993–2013) Then, FAOPenmanMonteith was used as the most valid method for determining the reference evapotranspiration after that crop coefficient was determined according to the cultivation method and eventually if this amount is multiplied to potential evapotranspiration determined from FAOpenmanMonteith method, the amount of the sugar beet evapotranspiration will determine. All of the calculation were run according to the daily average of the climate factors. Equation were determined by nonlinear regression method and SPSS software and in addition to the Blaney-Criddle equation parameters, GDD was entered in these equations and at last, the amount of the RMSE, R-2 and NS were calculated with

the amount of the ETc determined from FAO-penman

Monteith method . Also in a study that have been

carried out before, in order to define the amount of the sugar beet evapotranspiration, Blaney-Criddle method had higher precision among some equation such as Jensen-Heiz, Blaney-Criddle and tork so the preci sion of the determined equation and Blaney-Criddle method was compared. Finally, according to the deter mined results, the best equation for determining evapo transpiration was introduced for the whole region that had minimum error, maximum R2 and the best NS coefficient. The determined equation are as follows: where in all mentioned equations T is mean daily air temperature ($\circ$ C), U is wind speed (ms -1) , RH is rel ative humidity(-), P is the coefficient that is related to latitude, n is the actual of sunny hours and N is the maximum sunny, ETc is the crop evapotranspiration (mm/day) and GDD is the Growth Degree Day index.

2.2 FAO-PenmanMonteith equation

An equation that was presented theoretically on the basis of energy balance on a wet surface covered with plant is FAO-penmanMonteith equation.This method is as the most valid one for more actual determining of plants water requirement and was revised by FAO organization experts and by assuming a hypothetical reference crop with an assumed crop height of 0.12 m, a fixed surface resistance of 70 s m -1 and an albedo of 0.23 and leaf area index of plant is 4 times more than its height. Nowadays, FAO-PenmanMonteith equation is the principle of the water requirement calculation. Due to the absence of lysimeter data in long term and according to the FAO paper number 56 recommendation, this method is used as a standard method for evaluating other methods. FAO-PenmanMonteith equation is as follows: where ET0 is reference evapotranspiration (mm/day), Rn is net radiation at the crop surface (MJ m −2 day −1), G is soil heat flux density (MJ m −2 day −1), T is mean daily air temperature at 2 m height (∘ C), U 2 is wind speed at 2 m height (ms −1), es is saturation vapor pressure (kPa), ea is actual vapor pressure (kPa), esea is saturation vapor pressure deficit (kPa), is slope vapor pressure curve (kPa ∘ C-1), γ is psychometric constant (kPa ∘ C-1). 2.3 Blaney-Criddle equation One of the oldest methods for determining the potential evapotranspiration is Blaney-Criddle equation that its formula has been calibrated by Pruitt who is a professor of California University and this method was presented as follows for determining the grass evapotranspiration: where ET0 is the potential evapotranspiration (mm/day), P is the coefficient that is related to latitude, a and b are empirical parameters and T is the mean daily air temperature (∘ C). Amounts of “a” and “b” can be calculated as follows: where RH min is the minimum relative humidity, n is the actual of sunny hours and N is the maximum sunny, U is wind speed (ms −1). In former investigation that have been carried out in order to determine the daily potential evapotranspiration, when some methods were compared with FAOPenmanMonteith equation, Blaney-Criddle method had more precision proportional to other methods so this equation parameters were used for compose the new equations. In this method, the amount of sugar beet evapotranspiration was determined when sugar

beet crop coefficient is multiplied with potential evap

otranspiration and it was compared with determined

equations. In all of the mentioned equations above,

parameters that had been used are as follows: Where,

ETc is the evapotranspiration of plant, n is the number

of sunny hours (hr), P is daily average of light hours

in proportion to the total hours of lights in different

months of a year, T is average of daily temperature

(∘ C), GDD is growing degree day parameter, U is the

average of wind speed at 2 meter height of the surface, N is the number of the light hours in a day, RH is the amount of relative humidity along a day and “a” and “b” are climatic coefficient.

2.4 Fitness indexes

The amount of the NS, RMSE and R2 defines as fol lows, where Si and Qi is the calculated and actual amount respectively.

where, in 3 mentioned equation (14,15,16) S i is the amount of simulated values, S is the mean of simulated values, Q i is the observed values, Qis the mean of observed values, N is the number of data.

Growing degree day parameter (GDD):

Plant growing and crop producing is mainly effected by temperature, sun radiation, evaporate avail ability and temperature domain changes. Every plant needs specific amount of the temperature unite or degree days (GDD) to complete its growth and pro duce product. Any plant grows well in a finite domain of temperature changes that defines as cardinal tem perature. The amounts of the cardinal temperatures for sugar beet are 5 ∘ C and 30 ∘ C, respectively. The amount of the GDD defines as daily average tem perature minus minimum cardinal temperature. GDD cannot be smaller than zero or more than the cardinal temperature differences.

3 RESULT AND DISCUSSION

Determining the actual amount of evapotranspiration

or plant water demand is an important aspect in water

resource management in arid and semiarid areas. Since

there are not adequate lysimeter data and this method is so expensive and time-consuming in defining the amount of evapotranspiration, empirical methods have more importance. Specifying the amount of the evapotranspiration of reference crop with minimum climatic data and high precision and using the crop coefficient in order to achieve the amount of the evapotranspiration in standard and non-standard condition (potential and actual amount of the evapotranspiration) is one the most paramount problems that the researchers and experts of water department pay attention to it, specially. This problem has more importance in arid and semiarid areas and also in regions that there are not adequate climatic data. Several methods have been suggested in all over the world for determining the amount of the evapotranspiration that most of them needs to be calibrated or due to the lack of climatic data or requiring to the many inputs are not usable. Defining the exact amount of the evapotranspiration has high importance because of the water thrift. The aim of this study is defining a zonal equation with minimum climatic recorded data that can estimate the amount of the sugar beet evapotranspiration with high precision. In this study, we try to present an equation with more precision in order to define the amount of sugar beet water demand in Torbate-Jam so for comparing the amount of the defined equations precision with FAOPenmanMonteith method, as a standard method in determining the amount of the evapotranspiration, the amount of the RMSE, R2 and NS were calculated for every one of the defined equations. Each of these indexes has a domain of precision so as RMSE can vary between zero to extreme. This index represents the amount of the error so if the amount of this is near to zero then the amount of error is less. Changes domain of R2 is between zero and one. This index demonstrates the correlation of numeric so if the amount of the correlation coefficient is more, that equation is nearer to FAOPenmanMonteith. NS index was used for more precision. The Range of this index changes between one and minus extreme so if this index is nearer to one the mentioned equations is nearer to FAOPenmanMonteith method. The amount of the correlation between the amounts of the calculated evapotranspiration with FAOPenmanMonteith and every one of defined equations was demonstrated in fig 2. The average error between 9 presented equations is 9.33 and the amount of the correlation coefficient average is 0.646.As it is clear in fig 2, the maximum and minimum amount of the respective correlation belongs to third and seventh equation that equals 0.719 and 0.47, respectively but the amount of error is also important in determining the best equation. Third and sixth equations have the maximum and minimum amount of error that the amount of their error is 58.97 and 1.048, respectively. The maximum amount of NS coefficient belongs to first and eighth equation and Blaney-Criddle equation and the minimum amount of it belongs to third equation. As it is indicated before, third equation has the most amount of the correlation coefficient but the

Figure 2. The correlation lines between every equation and FAOPenmanMonteith equation.

Table 1. The amounts of correlation coefficient, error and NS of each method.

equation	R2	RMSE	NS
1	0.693	14.082	0.99
2	0.651	1.169	0.515
3	0.719	58.97	-1229
4	0.678	1.117	0.558
5	0.64	1.183	0.503
6	0.64	1.048	0.611
7	0.47	4.147	-5.08
8	0.64	1.21	0.99
9	0.683	1.05	0.99
(10) Blany-Criddle	0.657	1.22	0.99

amount of its error is high and equals to 59.67 and the amount of NS coefficient is not appropriate and equals to -1229 so although this equation has more correla tion rather than other equations but this equation has less precision.

4 CONCLUSION

For the first time, in this study, a new equation was presented by GDD parameter to define the amount of sugar beet evapotranspiration in Torbat-Jam. The amounts of the RMSE, R2 and NS coefficient was calculated by measuring ETc by FAOPenmanMonteith method that is the only acceptable method by FAO when there are not lysimeter data. In former investigation that has been done in Torbat-Jam to define the amount of sugar beet evapotranspiration by Hargreaves – Samani, Blany-Criddle, Tork and Jensen methods then defined results were compared with FAOPenmanMonteith as a standard method and results indicated that Blany-Criddle method has more precision rather than other ones. As it is explained above, the FAOPenmanMonteith method and defined methods were compared then the results were compared with Blany-Criddle method, results has been seen in table 1. According to these results, fourth equation has maximum amount of correlation coefficient and also minimum amount of error furthermore NS coefficient of this equation is near to 1 and it is good and equals to 0.99. It can be said that the results of fourth equation is better than Blany-Criddle method. This means that this equation can be used to increase the precision of calculation. The benefits of using this equation are according to this that the preci sion of evapotranspiration of this method is more than Blany-Criddle method, as it was mentioned before, Blany-Criddle method has the most precision in sugar beet water demand calculating, so if another suggested equation is used, there is no need to measure potential evapotranspiration and since the exact calculation of crop coefficient is so complicated. It does not require to define the crop coefficient of this plant in the dura

Morton F I (1994) Evaporation research, “A critical review and its lessons for the environmental sciences”, Crit. Rev. Environ. Sci. Tech. 24(3): 237-280.

Rahimain M H, Asadi H, The effect of water stress on quali tative and quantitative yield of sugar beet and determining the product function and crop coefficient of it. J. soil and water 12(10).

Samiei M, Hayati H, Mirzaei M (2013) Evaluating the various ways to calculate evapotranspiration based on radiation in Semnan synoptic station. J. Investigations of soil and water preservation. 19(3): 227-235.

Shahabi far M, Koochak zade M, Mohammad zade M, Mir lotfi S M(2004) Using geo statistical ways in order to determine the water demand of sugar beet in Tehran

province. 20(2):133–147. Sentelhas Paulo C,Terry J , Gillespie E S (2010) Evaluation of FAO Penman–Monteith and alternative methods for estimating reference evapo-transpiration with missing data in Southern Ontario. Agricultural Water Management, 97 (2010) 635–644. Su, Lijun Wang, Quanjiu Bai, Yungang)2013(An analysis of yearly trends in growing degree days and the relationship between growing degree day values and reference evapotranspiration in Turpan area, China. 113(3–4): 711–724. Wright JL (1982) New evapotranspiration crop coefficients. Am. Soc.of Civil Eng., J. of Irrig. And Drainage Div. 108 (IRI): 57–74.
Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Impact of rainfall variability on the sewerage system of Casablanca city, Morocco

L. Ennajem & D. Loudyi

Water and Environmental Engineering, University of Hassan II Casablanca,

Faculty of Science and Technology of Mohammedia, Morocco

ABSTRACT: Climate change is nowadays a major challenge facing mankind. Yet difficult to assess, the consequences of this phenomenon are multiple, irreversible and exceed the response capacity of ecosystems that may be damaged or permanently destroyed. Morocco has experienced a significant change in precipitation with increasing frequency and intensity of extreme events of drought and flooding type. In this article, the impact of rainfall variability on the Casablanca city sewage network is studied. The city is ranked as one of the most vulnerable coastal areas to floods risks due to rapid changes in urbanization, demography and also the change in the rainfall regime induced by climate change. The combination of these factors has increased the occurrence

of floods, causing damage to urban infrastructure and significant economic losses.

1 INTRODUCTION

In the context of climate change, there is variabil ity and changes in weather events, which results in increasing the number and intensity of extreme rain fall events. These changes affect the performance of urban drainage systems by increasing risks of flooding of drainage networks (Mailhot & Duchesne, 2005).Casablanca, the economic capital of Morocco, was particularly affected by the impact of these changes on its sewerage system. Indeed, the precipita tions of November 30, 2010 have exceeded the network discharge capacity and gave rise to floods that caused significant damage in this strategic city.

2 STUDY AREA: THE CITY OF CASABLANCA

Casablanca is the largest city in Morocco. It is the financial and economic capital of Morocco. The city houses the first financial center, the first port and the country's first university center. It is located on the Atlantic coast about 80 km south of the administrative capital Rabat. It is characterized by a semi-arid cli mate with average temperatures ranging from normal to 12.5 $\circ$ C in winter to 22 $\circ$ C in summer. The annual rainfall totals, characterized by high variability, have annual mean of 427 mm. They can reach values lower than 200 mm or sometimes exceed 800 mm.

3 RAINFALL VARIATIONS

In recent years, Morocco's climate has evolved towards

higher temperatures and lower annual total rainfall.

Projections for 2030 describe a warming of 1.3 ∘ C for the city of Casablanca and decreases of 6% to 20% of annual rainfall totals (Driouech et al. 2009). However, it is also envisaged that warmer temperatures and lower rainfall will be accompanied by more frequent and intense extreme precipitation events. Climate change influences in a major way the precipitations cycle at Casablanca, which results in an important variation in precipitation. (Figure 1) shows the evolution of annual rainfall in 16 years (1998 to 2014) (Driouech et al. 2009). 4 EVENTS OF NOVEMBER 27 TH , 2010 IN CASABLANCA Casablanca has experienced torrential rains in November 27 th , 2010 that reached 195 mm in 24 hours, which represents 50% of the precipitation annual average that was totally recorded in only one day. It was the most affected city by the overflowing of the Oued Bouskoura, during the day of November 30 th , 2010 (Fig. 2). The most affected areas by floods are the localities of El Gwassim, Rmel Lahlal, Oulad Malek, Nassime district, industrial district Sidi Maarouf, the exchange office, the Public Testing and Studies Laboratory (LPEE),the Hassania School, the Headquarter of the Phosphates Cherifien Office (OCP), and to a less extent, the Olympic Club of Casablanca, Oasis and Beausějour districts. Besides, many basic infrastructures that cross the Bouskoura River and surrounding homes were considerably affected. However, it is important to emphasize that the Bouskoura River is not the main cause of this dramatic situation experienced. The low capacity of stormwater pipes in urban areas was a major factor in the occurrence of floods.

Figure 1. The evolution of the annual rainfall for the period

1998–2014 (16 years).

Figure 2. Map of flooded areas in the Bouskoura River.

5 FACTORS INFLUENCING THE FLOODS OF

NOVEMBER 2010 IN CASABLANCA

5.1 Bouskoura river

Bouskoura river, located in the Chaouia watershed, arises downstream of the plain of Berrechid. At the entrance of the urban area, the stream flow cross sec tion is highly reduced, as the collector capacity is only 2 m 3 /s (compared to the decennal discharge of about 45 m 3 /s) (Word bank, 2011). The river natural course goes through the urban area of Casablanca, has become completely urbanized. In case of heavy rain, as in Jan uary 1996 and in November 2010, floods occur in the city center.

5.2 Urbanization

The rapid urbanization development upstream exist ing urban areas led to the emergence of new points of overflow. The insufficient downstream collectors are unable to absorb additional flows, and urban plan ning schemes are not properly designed to reduce

downstream flows of newly urbanized areas. Figure 3. Main collectors and sewerage system at the urban area of Greater Casablanca (source LYDEC, 2013). 5.3 Casablanca sewerage system Sewerage networks Management of Greater Casablanca is delegated to LYDEC (Lyonnaise des Eaux, the water private operator in Great Casablanca) since 1997. The concession contract covers an area of 120,000 hectares, of which 15,000 are urbanized. The network is a combined system in the older parts of the city and separate one in recent parts (Lydec, 2013). (Figure 3) shows the layout of the main collectors, the delimitation of the area where the network is combined and major drainage structures (storage basins, sewer outfalls and discharges in the ocean). 6 FLOOD DAMAGE The floods of the Bouskoura River occurred in January 1996 and November 2010, causing considerable damage to the Grand Casablanca (Fig. 4). The waters submerged the

Hassania School of engineers, the OCP headquarter and several buildings in El Jadida Road area. Three laboratories of LPEE were severely damaged. Long power cut and total panic in the streets have been experienced. Many roads were blocked, and cars trapped in underground garages because of the waters. The circulation was paralyzed and many schools were closed. 7 THE ADOPTED URBAN FLOOD OF THE BOUSKOURA RIVER 7.1 The River Bouskoura problematic Due to its location, The River Bouskoura represents a real threat for the metropolis. The riverbed is situated in the heart of the city of Casablanca. The river crosses the city from the East side to the West side, before flowing into Atlantic Ocean. In the last two decades, Casablanca has experienced rapid economic development in real estates, closing off the flow path of the river.

Figure 4. The damage of floods in November 2010 in El

Jadida road.

Nowadays, the pathway of the River Bouzkoura

goes along with: the road of El Jadida, Maarif,

Roudani Boulevard and La ligue Arab parc to reach

Houphouët-Boigny Boulevard.

The devastating floods of the river in 1995 and 2010,

causing major damages, and costing Casablanca’s bud

get millions of Dirhams. Led the city’s authorities to

implement a flood control channel to redirect excess

water to the ocean, Eager to protect the city from such

future disaster.

7.2 Super collector west project

The proposed solution is to install a 9 km control chan

nel in varying depths.With an open-air channel starting

from El Jadida Road ending into the ocean (Fig. 5).

After setting up this channel, this solution will have two characteristics, fist it is an artificial path river way to the Atlantic Ocean. Also it will support the wastew ater infrastructure in the west part of Casablanca who's experiencing rapid urbanization.

The model (flood control channel) consists of three parts:

– First part: it is 3 km open-air channel starting from the upstream of the Bouskoura River, this part is intended to carry the excess water when the flood

World Bank report and Egis BCEOM International/IAU IDF/BRGM. 2011. Phase 1: Évaluation des risques en situation actuelle et à l'horizon 2030 pour la ville de Casablanca. Projet: Adaptation au changement climatique et aux désastres naturels des villes côtières d'Afrique du Nord. World Bank report, GED 80823T, 261 p.

Lydec (Lyonnaise des Eaux, the water private operator in Great Casablanca). 2011. Activity Report. Lydec 2012. Schéma Directeur d'Assainissement. Aménagements de renforcement du réseau d'assainissement existant des zones urbanisées des communes de Casablanca et Mohammedia. Lydec 2014. Rainfall balance of the rainy season 2013–2014.
Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Implications of CMIP5 derived climate scenarios for discharge extremes of the Rhine

F.C. Sperna Weiland, M. Hegnauer & H. Van den Boogaard

Deltares, Delft, The Netherlands

H. Buiteveld & R. Lammersen

Rijkswaterstaat, Lelystad, The Netherlands

J. Beersma

KNMI, De Bilt, The Netherlands

ABSTRACT: In 2013 the IPCC published the 5th assessment report based on a new generation of global climate model simulations forced with Representative Concentration Pathways (RCP's). From these data KNMI constructed a set of four climate scenarios (van den Hurk et al, 2014). These scenarios tailored to the Rhine basin are used to assess the potential effects on discharge extremes in the river Rhine and their potential implications for flood protection design in the Netherlands. The assessment is conducted with the so-called Generator of Rainfall And Discharge Extremes (GRADE) which exists of a synthetic weather generator, a hydrological and a hydrodynamic model. Overall the instrument represents the probability of the observed discharge (extremes) well. According to all four scenarios (the probability of) discharge extremes in the Rhine will increase under future climate conditions.

1 INTRODUCTION

Within the Rhine river basin flood risk will likely increase due to ongoing economic developments, increasing population densities in flood prone areas and the impacts of climate change induced changes in rainfall and discharge extremes (Te Linde et al. 2010). In the Netherlands the current method for flood protection design requires the derivation of discharge extremes for return periods of 1000 up to 30,000 years. Since the available observed discharge records are too short to derive such extreme return levels, the extreme value analysis is based on very long synthetic dis charge time-series generated with the Generator of Rainfall And Discharge Extremes (GRADE, see Heg nauer et al. 2014). GRADE consists of two main components: a stochastic weather generator for the current climate (see subsection 2.4) and hydrological and hydrodynamic models (see subsection 2.2). To derive the probability distributions of extreme dis charges under climate change (Sperna Weiland et al., 2015) the synthetic rainfall and temperature series sim ulated with the weather generator are transformed to the future climate according to the recently issued KNMI'14 climate scenarios (van den Hurk et al. 2014; Lenderink & Beersma 2015) using an advanced delta change (ADC) method (van Pelt et al. 2012).

Previous studies reported on the use of weather generators for extreme value analysis under current climate conditions (Leander et al. 2005, Falter et al. 2015, Hundecha et al. 2009, Blazkova & Beven 2004). Here we assess discharge extremes under future climate conditions, considering the spatially varying change signal over the sub-catchments of the Rhine and Meuse basin derived from the RCM simulations used to construct the KNMI'14 climate scenarios (van den Hurk et al. 2014; Lenderink & Beersma, 2015) which in turn cover a large part of the range in the change of precipitation and temperature represented by the so-called CMIP5 GCM simulations used in the IPCC 5th assessment report (IPCC, 2013). 2 DATA AND METHODS This section describes the different components of GRADE for the Rhine basin. In a first step the hydrological and hydrodynamic models are described. After that we discuss the construction of synthetic meteorological time-series of 50,000 years that serve as input for the hydrological and hydrodynamic models to simulate discharge extremes for the current and future climate conditions. 2.1 The Rhine The river Rhine originates in the Swiss Alps, it flows along the Southern part of the boundary between France and Germany and continues through Germany

before it enters the Netherlands (Te Linde et al. 2011). The maximum observed discharge of 12,600 $m^3 s^{-1}$ was recorded in 1926 (Pinter et al. 2006). The Rhine is of great economic importance to agriculture, industry and navigation in Western Europe (Kwadijk & Rotmans 1995). The basin area is around 185,000 km^2 and has about 58 million inhabitants of which 10.5 million live in flood-prone areas (Te Linde et al. 2011, ICPR 2001). The major dikes along the Dutch part of the Rhine are designed with safety levels varying between 1/1000 to 1/30,000 years. This study focusses on the discharge gauge at Lobith near the Dutch-German border.

2.2 GRADE

The GRADE instrument consists of a logical chain of models, starting with a model for (synthetically) generation of precipitation and temperature, followed by a hydrological model that calculates natural river discharge from precipitation and finally a hydrodynamic model that propagates the discharge waves through the river channel. The synthetically generated precipita tion and temperature series have the correct frequency of extreme multi-day precipitation which are relevant for the generation of extreme river discharges.This has been checked in earlier papers (Buishand & Brandsma 2001; Buishand 2007; Schmeits et al. 2014). All relevant run-off processes including (ground) water storage, evaporation and snow accumulation and – melt are incorporated in the hydrological model. All hydraulic processes relevant for the propagation of discharge waves including dike overflows etc. are rep resented in the hydraulic model and finally the relevant climate change signal (from the climate change scenar ios) is incorporated in the long synthetic precipitation and temperature time series that are the input to the hydrological and hydraulic parts of GRADE for the simulation of future climate conditions. Below follows a description of the individual components.

2.3 Hydrological and hydrodynamic modelling

2.3.1 Hydrological modelling – HBV

River discharge is modeled with the HBV-96 model (Bergström 1992). The HBV model is a semi distributed conceptual hydrological model. It exists of an independent routine for snow melt and accumula tion following the degree-day method, a routine for groundwater recharge and a routine for actual evapo ration which are both functions of actual soil water storage and finally a routine for runoff generation which is released from a non-linear reservoir for fast runoff and a linear reservoir for base flow. Runoff is routed along the river with a simplified Muskingum approach. The model schematization used for the Rhine consists of 148 sub-basins, covering the com plete Rhine basin upstream of Lobith. It has recently been extended with individual sub-catchments for the large Swiss Lakes that have a considerable effect on the discharge of the Rhine (Hegnauer et al. 2014). The model has been re-calibrated following the GLUE methodology (Hegnauer et al. 2014; Winsemius et al. 2013) with performance criteria focussing on high discharges (i.e. Nash-Sutcliffe efficiency and Generalized Extreme Value Error) combined with the Relative Volume Error. The model requires precipitation and temperature as inputs. Potential evaporation is calculated based on a simple build-in temperature based correction of the sub-basin specific long-term monthly mean potential evaporation. 2.3.2 Hydrodynamic modelling SOBEK GRADE uses a hydrodynamic SOBEK-RE 1D model to simulate the propagation of the flood waves through the main stem of the Rhine. With the hydrodynamic model damping of the flood peaks caused by flooding of the Rhine in Germany can be accounted for. Hereto the so called retention option in Sobek-RE is used. With this option the flood areas behind the dikes are treated as a series of retention basins represented by buckets that are filled once the water in the river exceeds a certain level (i.e. approximately the level of the embankment). The location and the volume of the flooded areas (the retention basins) are pre-defined based on more detailed 2D models. The retention option enables SOBEK to mimic 2D flooding using a 1D approach which considerably decreases the computation time. 2.3.3 Correction for upstream flooding In extremely rare cases, the magnitude of the discharge may be that large that the water levels will exceed the top level of the embankments along the most downstream section of the River Rhine in Germany, between Wesel and Lobith. In these cases the schematization of the SOBEK model currently used is insufficient to simulate flood wave propagation including the effect of flooding accurately. As a result the model overestimates the discharge at Lobith. In several studies, based on 2D model simulations, the hydraulic maximum discharge of the Rhine at Lobith was estimated between 17,500–18,000 m^3/s (Paarlberg 2014; Lammersen et al. 2004) and this range is recently confirmed by an expert group (Hegnauer et al. 2015). Based on the results and data of these studies we assume that the maximum volume of water that can reach Lobith without exceeding the top of the German embankments is 18,000 m^3/s. From data analysis it is also assumed that the overflowing of the dikes along this stretch will start already when the discharge at Lobith exceeds 16,000 m^3/s, because locally the embankments are lower. We analyzed whether the amount of water overflowing the dikes can be limited by the capacity of the flood areas (called dike rings) and whether the dike overflow capacity could be a second limiting factor. Both did not pose any significant limitations. Based on the above discharges of more than 16,000 m^3/s are corrected, using equation 1, where Q is the calculated discharge which needs to be corrected, Q_0 is the lowest discharge where correction

starts (i.e. 16,000 m^3/s) and Q_L is the upper limit (i.e.

18,000 m^3/s).

2.4 Meteorological data

Gridded daily precipitation was taken from the

HYRAS 2.0 dataset (Rauthe et al. 2013). The dataset

has a resolution of 1 km^2, covers the period 1951

to 2006 and is based on 6200 precipitation stations.

The gridded time-series have been aggregated to the

HBV-Rhine sub-basins using Thiessen polygons.

Daily temperature time-series for the sub-basins

have been obtained by spatial aggregation (Thiessen's method) of the European gridded E-OBS dataset (Haylock et al. 2008).

2.5 Weather generator

The stochastic weather generator for the Rhine basin used within GRADE is based on nearest-neighbour resampling and produces very long rainfall and tem perature time series (with the same temporal resolution and sub-basins as the historical time series described in Section 2.3). Nearest neighbour (NN) resampling as a method of time series simulation was originally intro duced more or less independently by Young (1994) and Lall & Sharma (1996). Simultaneous multi-site and multi-variate NN resampling as used for the weather generator for the Rhine basin (Buishand & Brandsma 2001) has the advantage over parametric methods that no assumptions have to be made about temporaland spatial correlations and dependencies between precipitation and temperature. In addition, with resampling of daily (fields) of precipitation the probability distributions of extreme multi-day precip itation can more accurately be estimated than from classical extreme value analyses based on the (block) maxima or peaks over threshold (POT) of the histori cal multi-day precipitation amounts (Buishand 2007). The latest version of the Rainfall generator for the

Rhine basin is described in Schmeits et al. (2014).

With this version 50,000-year time series were simu

lated representative of the current climate. In addition

these 50K-year time series were transformed to the

future climate as described in the following section,

to obtain, together with the other components of

GRADE, discharge scenarios for the future climate.

2.6 Development of local climate scenarios

The first generation of climate change scenarios that

were combined with GRADE were the so-called

KNMI'06 climate change scenarios (Hurk et al. 2006).

Although the KNMI'06 scenarios were specifically

developed for the Netherlands, these scenarios were

applied to the whole Rhine basin to force GRADE and to derive future discharge scenarios consistent with the KNMI'06 climate scenarios. In the RheinBlick2050 project (Görgen et al. 2010) the GRADE methodology was combined with an ensemble of (bias-corrected) Regional Climate Model (RCM) simulations to derive future (extreme) discharge scenarios for various locations along the Rhine. In 2014 the KNMI'14 scenarios (Hurk et al. 2014; Lenderink et al. 2014; Lenderink & Beersma 2015) were issued as a successor of the KNMI'06 scenarios. This time not only scenarios were developed for the Netherlands but also specific ones for the Rhine and Meuse basins (Sperna Weiland et al. 2015; Lenderink & Beersma 2015). In this paper scenarios for the change in (extreme) discharge of the Rhine are derived by combining GRADE with the KNMI'14 climate scenarios tailored for the Rhine basin. The KNMI'14 scenarios were derived from an ensemble of simulations with the EC-Earth global climate model downscaled with the regional climate model RACMO2 (Lenderink et al. 2014). The range of climate change that the set of KNMI'14 climate scenarios spans is determined by the CMIP5 global climate model simulations also used for the fifth IPCC assessment report (IPCC 2013). 2.6.1 CMIP5 CMIP5: the Coupled Model Intercomparison

Project (CMIP) phase 5 is the most recently completed collaboration for running and comparing global coupled ocean-atmosphere general circulation models (Taylor et al. 2012). Within the CMIP5 the global climate models are forced with a common set of four so-called Representative Concentration Pathways (RCPs), representing different pre-defined levels of global population growth, economicand technological developments and mitigation policies. The numbers that are associated with the RCPs (2.6, 4.5, 6.0 and 8.5) represent radiative forcing in 2100 i.e. the additional energy (in Watts/m 2) that is systematically added to the earth-system as a result of increased greenhouse gas concentrations. CMIP5 forms the basis for the climate model projections described in the 5th IPCC assessment report. These CMIP5 simulations are used as a reference for the KNMI'14 climate scenarios. 2.6.2 KNMI'14 As said, the CMIP5 simulations are the reference for the KNMI'14 scenarios. However, the low emission scenario RCP2.6, which is mainly an emission scenario developed for mitigation, is neglected. Of the remaining ~200 CMIP5 simulations the percentiles of the changes in seasonal mean precipitation and temperature over the Rhine basin are determined. The initial aim was that the set of four KNMI'14 scenarios represent the 10–90% of the CMIP5 spread for summer and winter changes in seasonal means. With a fifth scenario for the Rhine and Meuse basins (Lenderink & Beersma 2015) we managed that for all scenarios considered the KNMI'14 scenarios span at least 50–80% of the CMIP5 spread. Each of the individual KNMI'14

Figure 1. Overview of the modelled and measured annual

maxima (1951–2006) for the Rhine at Lobith. The dashed

lines indicate the line of equality (blue) with its +1000 m 3 /s

and −1000 m 3 /s band.

scenarios is represented by a selected specific sample

from an ensemble of EC-Earth/RACMO2 simulations

(Lenderink et al. 2014). The change statistics in these

scenarios are used to transform the long synthetic time

series of precipitation and temperature, representative

for the current climate, for each of the sub-basins –

simulated with the stochastic weather generator (see Section 2.4.) – to long synthetic time series representative for the future climate using the Advanced Delta Change (ADC) method described in Van Pelt 0 (2012) and Sperna Weiland et al. (2015). This method allows the extreme (multi-day) precipitation to change differently from the mean precipitation and also accounts for the spatial gradients of climate change over the Rhine basin, the latter in contrast to the KNMI'06 scenarios used earlier for the Rhine basin.

3 RESULTS

3.1 Evaluation of the hydrological and hydrodynamic model against observations

In a brief evaluation of the hydrological and hydrodynamic components of GRADE we compare the modelled annual maxima with measurements. Figure 3.1 shows that overall the modelled annual maxima compare reasonably well with observations. The large deviation of two flood events above the red dotted line (+1000 m^3/s) is associated with a strong overestimation of the discharges upstream at Maxau which is caused by the absence of retention modeling for the area between Basel and Maxau.

3.2 Future discharge extremes

In Fig. 3.2 the cumulative distributions of the annual maximum 10-day precipitation (left) and discharge modelled with HBV (right) are shown for 2050 and 2085. It can be seen that both extreme discharges and extreme 10-day precipitation increase with respect to the current climate (black lines). For the higher return periods (e.g. above 100 years), the increase in the dis charge is around 10–15%. The same can be observed in the 10-day precipitation – frequency curves. The Figure 2. Cumulative (probability) distributions of the maximum 10-day precipitation in the Rhine basin in the winter half year (left) and of the annual discharge maxima at Lobith (right) for the four KNMI'14 scenarios for 2050 (top) and 2085 (bottom) based on the hydrological model results (HBV), without the effect of upstream flooding. The black curve represents the reference situation (i.e. the current climate). Figure 3. Discharge – frequency curves for the Rhine at Lobith for all scenarios and years, based on the hydrodynamic model results (Sobek), with the effect of upstream flooding. In grey the results without the correction for the flood areas along the German stretch upstream of Lobith are shown. spread in the scenarios is still relatively small for 2050. The reason for the small spread is found in the climate scenarios. In 2050 the spread between the four scenarios in winter is about 13% for the change in mean precipitation (Lenderink & Beersma 2014). Apparently the spread in the change of the extreme 10-day precipitation events in the winter half year in the four scenarios is considerably smaller, only about 5%. The graphs in Figure 3.3 contain the discharge frequency curves obtained after flood propagation through the hydrodynamic model and correction for upstream flooding in the Lower Rhine section for 2050 (left) and 2085 (right). In grey the results without the correction for the flood areas along the German stretch upstream of Lobith are shown. According to the HBV simulations, where upstream flooding is not yet considered, the discharges in 2085 for for exam ple the 1250-year event range between 18,700 and 22,300 m^3/s. Whereas the reference estimate of the 1250-year event was approximately 16,900 m^3/s. Upstream flooding in the Rhine reduces the highest discharges at Lobith significantly. In 2085 the corrected 1250-year discharge ranges between 14,950 and 17,100 m^3/s for the G_H and W_H scenarios. The reference estimate of the 1250-year discharge is 14,380 m^3/s.

4 DISCUSSION AND CONCLUSIONS

The with the hydrological and hydrodynamic model simulated annual maxima are similar to observed which indicates that GRADE can be used for the simulation of discharge extremes for the Rhine and future changes therein.

The part of the modeling chain that gives rise to most discussion is the correction of discharges above 16,000 m^3/s for upstream flooding. This could (and should) be improved by full 2D hydrodynamic modeling of flooding in the Lower Rhine. Currently a 2D hydrodynamic model is being developed for this river stretch.

All KNMI'14 scenarios project increases in discharge extremes for the river Rhine under future climate conditions hereby posing increased require ments for flood protection design to maintain the current safety standard.

Bergström, S. 1992. The HBV model – its structure and applications. SMHI RH No 4. Norrköping. 35 pp.

Blazkova, S. & Beven, K. 2004. Flood frequency estimation by continuous simulation of subcatchment rainfalls and discharges with the aim of improving dam safety assess ment in a large basin in the Czech Republic, J. Hydrol. 292: 153–172.

Buishand, T.A. 2007. Estimation of a large quantile of the distribution of multi-day seasonal maximum rainfall: the value of stochastic simulation of long-duration sequences. Climate Res. 34(3): 185–194, doi:10.3354/cr00701.

Buishand, T.A. & Brandsma, T. 2001. Multi-site simulation of daily precipitation and temperature in the Rhine basin by nearest-neighbor resampling. Water Resour. Res. 37: 2761–2776.

Falter, D., Schröter, K., Dung, N.V., Vrogushyn, S., Kreibich, H., Hundecha, Y., Apel, H. & Merz, B. 2015. Spatially coherent flood risk assessment based on long-term continuous simulation with a coupled model chain, J. of Hydrol. 524: 182-193, http://dx.doi.org/10.1016/j.jhydrol.2015.02.021.

Görgen, K., Beersma, J., Brahmer, G., Buiteveld, H., Caram bia, M., de Keizer, O., Krahe, P. & Nilson, E. 2010.Assess ment of Climate Change Impacts on Discharge in the Rhine River Basin: Results of the RheinBlick2050 Project. (PDF), International Commission for the Hydrology of the Rhine Basin (CHR), Lelystad, 2010, ISBN 978-90-70980-35-1, S. 229, CHR report I-23 (Zugriff am April 1, 2010). Hegnauer, M., Beersma, J.J., Van den Boogaard, H.F.P., Buishand, T.A. & Passchier, R.A. 2014. Generator of Rainfall and Discharge Extremes (GRADE) for the Rhine and Meuse basins: Final report of GRADE 2.0. Deltares report 1209424-004-ZWS-0018, Delft, The Netherlands. Hegnauer, M., Kwadijk, J. & Klijn, F. 2015. The plausibility of extreme high discharges in the river Rhine. Deltares report 1220042-004-ZWS-0008. Hundecha, Y., Pahlow, M. & Schumann A. 2009. Modeling of daily precipitation at multiple locations using a mixture of distributions to characterize the extremes. Water Resour. Res. 45(12): W12412. http://dx.doi.org/10.1029/2008WR007453. Hurk, B.J.J.M. van den, Klein Tank, A.M.G., Lenderink, G., van Ulden, A.P., Van Oldenborgh, G.J., Katsman, C.A., Van den Brink, H.W., Keller, F., Bessembinder, J.J.F., Burgers, G., Komen, G.J., Hazeleger,W. & Drijfhout, S.S. 2006. KNMI Climate Change Scenarios 2006 for the Netherlands. KNMI-publication WR-2006-01, 30/5/2006, pp82, KNMI, De Bilt, The Netherlands. Hurk, B. van den, Siegmund, P., KleinTank,A. (Eds),Attema, J., Bakker, A., Beersma, J., Bessembinder, J., Boers, R., Brandsma, T., van den Brink, H., Drijfhout, S., Eskes, H., Haarsma, R., Hazeleger, W., Jilderda, R., Katsman, C., Lenderink, G., Loriaux, J., Van Meijgaard, E., Van Noije, T., Van Oldenborgh, G.-J., Selten, F., Siebesma, P., Sterl, A., deVries, H.,Van Weele, M., de Winter, R. & van Zadelhoff, G.-J. 2014. KNMI'14: Climate Change scenarios for the 21st Century – A Netherlands perspective. KNMIpublication WR-2014-01, 26/5/2014, pp120, KNMI, De Bilt, The Netherlands. ICPR, 2001. Atlas of flood danger and potential damage due to extreme floods of the Rhine. International Commission on the Protection of the Rhine, Koblenz, Germany, available at: http: //www.iksr.org/, 2001. IPCC, 2013. Climate Change 2013: The Physical Science Basis. Contribution ofWorking Group I to the FifthAssessment Report of the Intergovernmental Panel on Climate Change [Stocker, T.F., D. Qin, G.-K. Plattner, M. Tignor, S.K. Allen, J. Boschung, A. Nauels, Y. Xia, V. Bex and P.M. Midgley (eds.)]. Cambridge University Press, Cambridge, United Kingdom and New York, NY, USA, 1535pp, doi:10.1017/CBO9781107415324. Kwadijk, J.C.J. & Rotmans, J. 1995. The impact of climate change on the river Rhine: a scenario study. Climatic Change 30(4): 397–425. Lall, U. & Sharma, A. 1996. A nearest neighbour bootstrap for resampling hydrologic time series. Water Resources Research

32: 697–693. Lammersen, 2004. Grensoverschrijdende effecten van extreem hoogwater op de Niederrhein. ISBN 9036956390, RIZA, Arnhem. Leander, R., Buishand, T.A., de Wit, M.J.M. & Aalders, P. 2005. Estimation of extreme floods of the river Meuse using a stochastic weather generator and rainfallrunoff model. Hydrolog. Sci. J. 50: 1089–1103, doi: 10.1623/hysj.2005.50.6.1089. Lenderink, G., van den Hurk, B.J.J.M., Klein Tank, A.M.G., van Oldenborgh, G.J., van Meijgaard, E., de Vries, H. and Beersma, J.J. 2014. Preparing local climate change scenarios for the Netherlands using resampling of climate model output. Environmental Research Letters 9(11): 115008, doi:10.1088/1748-9326/9/11/115008.

Lenderink, G. & Beersma, J.J. 2015. The KNMI'14 W H ,dry scenario for the Rhine and Meuse basins. KNMI publication WR-2015-02, 12/2015, pp24, KNMI, De Bilt, The Netherlands.

Paarlberg, A. 2014. GRADE Niederrhein: Dijkoverstroming versus dijkdoorbraak. HKV report PR2942.10, HKV, Lelystad, The Netherlands.

Pinter, N., Van der Ploeg, R.R., Schweigert, P. & Hoefert, G. 2006. Flood magnification on the river Rhine, Hydrol. Processes 20: 147–164.

Schmeits, M.J., Wolters, E.L.A., Beersma, J.J. & Buis hand, T.A. 2014. Rainfall generator for the Rhine basin: Description of simulations using gridded precipitation datasets and uncertainty analysis. KNMI-publication PUBL-186-VII, 1/2014, pp29, KNMI, De Bilt, The Netherlands.

SpernaWeiland, F., Hegnauer, M., Bouaziz, L. & Beersma, J.J. 2015. Implications of the KNMI'14 climate scenarios for the discharge of the Rhine and Meuse; comparison with

earlier scenario studies. Report nr. 1220042-000-ZWS

004, Deltares, Delft, The Netherlands. Taylor, K.E., Stouffer, R.J. and Meehl, G.A. 2012. An overview of CMIP5 and the experiment design. Bull.Amer. Meteor. Soc. 93: 485–498. doi: http://dx.doi.org/10.1175/ BAMS-D-11-00094.1 Te Linde, A.H., Bubeck, P., Dekkers, J.E.C., de Moel, H. & Aerts, J.C.J.H. 2011. Future flood risk estimates along the river Rhine. Nat. Hazards Earth Syst. Sci. 11: 459–473. Te Linde, A.H., Aerts, J., Bakker, A.M.R. & Kwadijk, J.C.J. 2010. Simulating low probability peak discharges for the Rhine basin using resampled climate modeling data,Water Resources Research 46, doi:10.1029/2009WR007707. Young, K.C. 1994. A multivariate chain model for simulating climatic parameters from daily data. Journal of Applied Meteorology 33: 661–671. Winsemius, H.C., Van Verseveld, W., Weerts, A. & Hegnauer, M. 2013. Generalised Likelihood Uncertainty Estimation for the daily HBV model in the Rhine Basin. Deltares report 1207771-003-ZWS-0018. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

How will be future rainfall IDF curves in the context of climate change?

H. Tabari, P. Hosseinzadehtalaei & P. Willems

Hydraulics Division, Department of Civil Engineering, KU Leuven, Belgium

S. Saeed, E. Brisson & N. Van Lipzig

Department of Earth and Environmental Sciences, KU Leuven, Leuven, Belgium

ABSTRACT: The design statistics for water infrastructures are typically derived from rainfall intensity-

duration-frequency (IDF) curves which compound frequency and intensity aspects of rainfall events for different

durations. Current IDF curves are constructed based on historical time series, with an underlying temporal sta

tionarity assumption for the probability distribution of extreme values. However, climate change casts doubt on the validity of this assumption due to ongoing and projected changes in the intensity and frequency of extreme rainfall. In this study, IDF curves for historical periods obtained from the convection permitting CCLM model with spatial and temporal resolutions of 2.8 km and 15 minutes and an ensemble of climate models (CMIP5) are validated based on observations-based curves. After this validation, future climate IDF relationships are obtained based on a quantile perturbation approach. It is concluded that the sub-hourly precipitation intensities at 15 and 30 minutes in the IDF curves derived from the CCLM 2.8 km model underestimate the observed extreme rainfall intensities. For the daily intensities, less deviation is observed for both the CCLM and the CMIP5 GCM runs. Future climate projections show potentially strong changes in extreme rainfall intensities, making the historical climate based IDF design standards unsuitable for the future extreme events.

1 INTRODUCTION

Urban drainage systems, stormwater infrastructure, culverts and other hydraulic structures are designed based on specific design storms derived from rainfall Intensity–duration–frequency (IDF) curves. The IDF curves quantify the frequency of occurrence of a storm with a specific intensity at different durations. The curves are traditionally derived from historical rainfall records by making a temporal stationarity assumption of rainfall series, implying that the intensity and frequency of historical rainfall extremes remain unchanged for the future. However, the Fifth Assess ment Report from the Intergovernmental Panel on Climate Change (IPCC 2013) reported that the fre quency and intensity of extreme rainfall would likely increase in a warmer climate. Accordingly, an increase in these characteristics of intense rainfall has been observed over most land areas in the late 20th century (e.g., Frich et al. 2002; Madsen et al. 2009; Tabari et al. 2014). Hence, as the pattern of extreme events shifts, the expected changes and uncertainty have to be incor porated in the IDF curve construction to reduce the risk of malfunction or failure of the current infrastructures in the future. In order to examine how the intensity and frequency of heavy rainfall events will change in the future, climate model projections under current and future greenhouse gas scenarios can be applied.

Global climate models (GCMs) have been predom inantly used for estimation of future modification in intense rainfall under increased greenhouse gas conditions (e.g., Mpelasoka & Chiew 2009; Tabari et al. 2015; Asokan et al. 2016). However, the spatial resolution of GCMs is hundreds of kilometers which poses limitations to the explicit simulation of mesoscale processes involved during extreme precipitation as well as to the representation of the land surface features (Frei et al. 2006; Hassanzadeh et al. 2014). The processes that cannot be resolved in horizontal grid spacing of GCMs are parameterized, which is a source of large bias and uncertainty in the simulations (Prein et al. 2013; Kendon et al. 2014; Olsson et al. 2015). A more trustworthy representation of these processes and features is provided at finer spatial resolutions of regional climate models (RCMs), dynamically derived from either GCMs or reanalysis data. The spatial resolution of RCMs is typically between 50 and 12 km, for instance 50 km for RCMs implemented and simulated in the project PRUDENCE (Christensen and Christensen 2007) and NARCCAP (Mearns et al. 2009), 25 km in ENSEMBLES (van der Linden and Mitchell 2009) and 12 km in EURO-CORDEX (Giorgi et al. 2009). Even if the spatial resolution of RCMs is much higher than that of GCMs, the grid size is still too large to adequately represent convective rain which is primary importance for urban flood risk analysis. During the past few years, considerable efforts have been made to develop climate models with spatial resolutions less than about 4 km (namely, convection permitting model) at which deep convective phenomena are sufficiently resolved, and orography and land surface are represented more

realistically (e.g., Mahoney et al. 2013; Attema et al.

2014; Kendon et al. 2014; Brisson et al. 2015).

Next to the need of high spatial resolution of design storms, precipitation data with durations of less than a day and even less than an hour are required for urban hydrological applications (Willems et al. 2012; Gregersen et al. 2013). Such high temporal resolution data are rarely provided by GCMs and even if this is the case, the data are associated with large biases. Even though computationally intensive, RCMs are generally expected to provide more realistic sub-hourly rainfall series (Kuo et al. 2014). A number of studies have per formed to develop future IDF curves based on either statistically downscaled rainfall series from GCMs or RCM outputs (e.g., Mailhot et al. 2007; Willems & Vrac 2011; Wang et al. 2013; Rodriguez et al. 2014; Mirhosseini et al. 2015), but there are only a few instances of sub-hourly convection permitting model use in the literature.

To address both fine spatial and temporal needs, we use sub-daily precipitation outputs from the COSMO CLM (CCLM) model with spatial scale of 2.8 km for future IDF curves construction. In addition to the CCLM model, the IDF relationships are analyzed based on the precipitation results from an ensemble of the Coupled Model Intercomparison Project of

Table 1. Climate models and number of runs per model used in this study (194 runs).

	Resolution		Number of runs				
	Longitude	Latitude					
Model	(degree)	(degree)	Historical	RCP2.6	RCP4.5	RCP6.0	RCP8.5
ACCESS 1.0	1.9	1.3	1	-	1	-	1
ACCESS 1.3	1.9	1.3	1	-	1	-	1
BCC-CSM1.1	2.8	2.8	3	-	-	-	-
BCC-CSM1.1(m)	1.1	1.1	1	1	1	1	1
BNU-ESM	2.8	2.8	1	1	1	-	1
CanESM2	2.8	2.8	5	5	5	-	5
CMCC-CM	0.8	0.8	1	-	1	-	1
CMCC-CMS	1.9	1.9	1	-	1	-	1
CMCC-CESM	3.7	3.8	1	-	-	-	1
CNRM_CM5	1.4	1.4	1	1	1	-	1
CSIRO-MK3.6.0	1.9	1.9	10	10	10	10	10
FGOALS-G2	4.7	2.8	1	1	1	-	1
GFDL-CM3	2.5	2.0	2	1	1	1	1

GFDL-ESM2G 2.5 2.0 1 1 1 1 1

GFDL-ESM2M 2.5 2.0 1 - 1 1 1

GISS-E2-H 2.5 2.0 2 - - - -

GISS-E2-R 2.5 2.0 3 - 2 - -

HADGEM2-CC 1.9 1.3 3 - 1 - 1

HADGEM2-ES 1.9 1.3 1 1 - - -

INM-CM4 2.0 1.5 1 - - - 1

IPSL-CM5A-LR 3.8 1.9 4 - 3 1 3

IPSL-CM5A-MR 2.5 1.3 1 1 1 1 1

IPSL-CM5B-LR 3.8 1.9 1 - 1 - 1

MIROC-ESM 2.8 2.8 1 1 1 1 1

MIROC-ESM-CHEM 2.8 2.8 1 1 1 1 1

MIROC5 1.4 1.4 3 2 1 1 3

MPI-ESM_LR 1.9 1.9 1 1 1 - 1

MPI-ESM_MR 1.9 1.9 1 1 1 - 1

MRI-CGCM3 1.1 1.1 1 1 1 1 1

NorESM1-M 2.5 1.9 3 1 1 1 1 the World Climate Research Programme - Phase 5 (CMIP5) GCMs. To the best of our knowledge, this is the first time that future IDF curves are developed based on such high spatial (i.e., 2.8 km) and temporal (i.e., hourly) resolution climate model projections and such large ensemble of GCMs (around 200 runs). 2 MATERIALS AND METHODS 2.1 Data This study makes use of the new GCM outputs for daily precipitation from the CMIP5 multi-model experiment. In the dataset, future precipitation projections for the 21st century, which extend to 2100, are forced by prescribed levels of total radiative forcing, referred to as Representative Concentration Pathways (RCPs), determined by a cumulative measure of greenhouse gas emissions, land use and air pollution. Four greenhouse gas concentration scenarios namely RCP2.6, RCP4.5, RCP6.0 and RCP8.5 were used in this study. In total, the precipitation data for 194 GCM runs (historical control run, and RCP based scenario runs) from 30 climate models with spatial resolutions between 89 km and 523 km are analyzed. Table 1 lists the name of the models and the number of runs per model.

All historical control runs were obtained from the CMIP5 database for the period 1961–1990, and all scenario runs were considered for the future period 2071–2100.

Next to the CMIP5 GCMs, sub-daily precipita tion data from a high-resolution climate model named CCLM were utilized. This model is a non-hydrostatic limited area climate model developed by the cli mate limited-area modeling (CLM) community. The domain for the model simulations has a large size (192 × 175 gridpoints) in order to exclude the spa tial spin-up effects (Brisson et al. 2015). A three-step nesting strategy is applied with the driving data (either from ERA-Interim or EC-EARTH) forcing a domain at 25 km grid mesh size, which in turn forces an inte gration at 7 km grid mesh size. The integration at 2.8 km grid spacing is forced with the output from the 7 km integration. Model integrations were performed for the period 2001–2010, and a thorough evaluation of decadal statistics of precipitation, temperature and cloud characteristics was recently performed (Brisson et al. 201X). The CCLM driven by EC-EARTH was performed for the period 2000–2010 and 2060–2069 using the RCP4.5 emission scenario.

For all GCM and CCLM runs, precipitation data were extracted for the pixel covering Uccle station in Central Belgium.This was done as high quality 10-min precipitation data measured with the same instrument are available at this station for more than a century, of which the data for the period 1961–2010 are used in this study. In addition to the 10-min station obser vations, daily E-OBS gridded data for 27.8 km and 55.7 km were used. These gridded data were aggre gated to larger pixels of 167 km and 501 km for a comparison with gridded CMIP5 GCMs.

2.2 Methods

The IDF curves for 1-month, 1-year and 10-year return periods and for durations from 10 minutes up to one month were developed for the control and sce nario periods of the climate models as well as the observations. The IDF curves were derived based on POT extreme value statistics after calibration of two-component exponential distributions, following Willems (2000).

To project the future IDF relationships, a delta change approach was applied using the following steps:

1. Derive design precipitation of different return peri

ods and durations for the control period;

2. Derive design precipitation of the same return peri

ods and durations as those for the control period for

the scenario period;

3. Compute change factors as the ratio of design pre

cipitation for the scenario period over that for the

control period with the same return period and

durations;

4. Multiply the change factors to the design precipita

tion of the observations with the same return period

and duration. The procedure is summarized as the following relationship: where P (F ,t) , P (O,t) , P (S,t) and P (C,t) are design precipitation of return period t for the future, the observations, and the scenario and control periods, respectively. 3 RESULTS AND DISCUSSION 3.1 Validation of climate models-driven IDF curves The IDF curves using the CCLM model as well as its driving GCM (EC-EARTH) and reanalysis (ERAInterim) were constructed for the historical period (2001–2010). Figure 1 shows these IDF curves with reference to the precipitation intensities from the station and E-OBS pixel data over Uccle location (Central Belgium). For sub-hourly precipitation intensities (15 and 30 minutes), which are typically used for sewer and drainage system design, the CCLM model of 2.8 km resolution tends to underestimate the precipitation intensities. For instance, for a storm of 10-year return period and 15-min duration, this underestimation can be up to 63 mm/h. Although this underestimation may be partially due to spatial scale difference, however, in reality the IDF curves from station observations (and not gridded observations) are used for the design of hydraulic structures. For sub-daily durations (hourly, 3 hourly and 6 hourly), precipitation intensities are underestimated by almost all the CCLM model runs except for some 2.8 and 7 km runs. In the case of daily to monthly durations, the precipitation intensities simulated by the models are very close to each other. Nevertheless, it can be concluded from the IDF plot that the precipitation intensities are overestimated by the driving ERA-Interim reanalysis data and underestimated by the driving EC-EARTH GCM. The IDF curves were also developed using the 58 control runs of the CMIP5 GCMs for the period 1961– 1990 (Fig. 2). It was difficult to include the IDF curves for the gridded data with different pixel sizes equal to that of each GCM, so the IDF curves for only the upper and lower limit of pixel sizes were added to the plot in Figure 2. As shown, most of the simulated IDF curves by the CMIP5 GCMs are within the range provided by the gridded data. The GCMs' IDF curves underestimate those based on the station observations which are used in practice for hydraulic infrastructure design. 3.2 Future IDF curves After validation of the IDF curves estimated by the climate models for the control period, the future IDF curves are projected based on the quantile perturbation approach (Willems & Vrac, 2011; Ntegeka et al., 2014). Instead of using this perturbation downscaling method for perturbing precipitation time series to develop the IDF curves for the future (e.g. Willems,

Figure 1. Baseline IDF curves using the CCLM model and its driving GCM and reanalysis versus corresponding IDF curves

obtained using point and pixel observations at Uccle for the historical period 2001–2010.

Figure 2. Baseline IDF curves using the CMIP5 GCMs versus corresponding IDF curves obtained using point and pixel

observations at Uccle for the historical period 1961–1990.

2013), we perturbed design precipitation considering

three aspects of precipitation (i.e., intensity, duration

and frequency). The change factors in IDF relation

ships between the control (2000–2001) and scenario (2060–2069) runs of the CCLM model for different return periods and aggregation levels are presented in Figure 3. Both increasing and decreasing precipitation intensities are projected by the CCLM model.

Figure 3. Relative changes in IDF relationships between the control (2000–2001) and scenario (2060–2069) runs of the

CCLM model for different return periods and aggregation levels.

The most noticeable change is a 97% increase in hourly precipitation intensity for the 10-year return period by the CCLM 2.8 km model. The 46% reduction in daily precipitation intensity by the CCLM 7 km model and 34–35% increase in daily and 6-h precipitation inten sities by the CCLM 25 km model are the other notable changes in IDF relationships.

The change factors for the ensemble of the CMIP5 GCMs for all RCPs (RCP2.6, RCP4.5, RCP6.0 and RCP8.5) indicate an increase in extreme precipitation intensities of 1and 10-year return periods for all dura tions (Fig. 4). For the smaller 1-month return period, the increase is observed only for 1and 5-day dura tions, while the other aggregation levels show a slight decrease. For the higher precipitation intensities (T > 1 year), the increment in design precipitation increases with decreasing duration and increasing return period. Based on the 95th percentile (median) of change fac tors by the GCMs, which is shown by the upper whisker (red line inside the box) in the box-plot in Figure 4, the precipitation intensity of 10-year return period and daily duration increases by 48% (15%).

The future IDF curves obtained based on the CCLM model results downscaled for the study region for three return periods (T = 1 month, 1 year and 10 years) and ten durations (from hourly to monthly) are shown in Figure 5. The plot shows that future IDF curves

are expected to change with the future climate. The

most noticeable change is seen for the hourly precipi

tation intensity of 10-year return period by the CCLM 2.8 km. Around 100% increase by this model implies that a hydraulic structure designed for storms of a 10year return period will significantly be underdesigned for the future extreme events. The future IDF curves for the end of the 21st century (2071–2100) using the CMIP5 GCMs versus the existing IDF curves (1961–1990) based on Uccle station observations are for each RCP scenario separately plotted in Figure 6. As one can see, the precipitation intensities for the future IDF curves are predominantly higher than their corresponding values for the existing IDF curves. For instance, for a 10-year return period with daily duration, a high percentage precipitation intensity increase approximately by 37%, 35%, 50% and 64% relative to the existing IDF curves is projected for the future IDF curves for the RCP2.6, 4.5, 6.0 and 8.5, respectively. This indicates that the current hydraulic structures and other water infrastructure will not be able to cope with more intense and frequent extreme events in the future. The strongest increase in extreme precipitation obtained in this study for Central Belgium using the CMIP5 GCMs is much higher than that (about 30% increase) reported by Willems & Vrac (2011) based on a set of 17 ensemble runs from the ECHAM5 general circulation model. Results are more comparable but again higher for the RCP8.5 scenario than the maximum increase of 50% reported by Willems (2013) based on an older (CMIP3) generation ensemble of 44 RCM and 69 GCM runs.

Figure 4. Relative changes in IDF relationships between the control (1961–1990) and scenario (2071–2100) runs of the

CMIP5 GCMs for different return periods and aggregation levels, combining all RCPs.

Figure 5. Future IDF curves (2060–2069) after quantile perturbation based on the CCLM model versus the historical climate

IDF curves (2001–2010) at Uccle.

Figure 6. Future IDF curves (2071–2100) after quantile perturbation based on the CMIP5 GCMs versus the historical climate

IDF curves (1961–1990) at Uccle.

4 CONCLUSIONS

This study focused on validation and future projection of IDF curves obtained from the CCLM model and a large ensemble of CMIP5 GCMs, with precipitation durations ranging between 15 minutes and one month. The future IDF curves were developed by applying climate change factors on extreme rainfall quantiles to existing IDF curves that were based on raingauge observations.. This was done to the respective dura tions and return periods, in the framework of a quantile perturbation downscaling approach. The results show a clear tendency of the expected extreme precipita tion intensities for the future to increase. Considering the higher intensities of precipitation (T > 1 year), the amount of increase is higher for smaller time scales and larger return periods. The precipitation intensity with hourly time scale and 10-year return period may increase up to about 100%. Furthermore, the increase in the design storm intensities as derived from the CMIP5 ensemble increases with the CO_2 concentra tions in the emission scenarios, ranging from 37% in the RCP2.6 scenario to 64% in the RCP8.5 scenario. The results of this study indicate that the current IDF curves are not sufficient to represent future precipita tion patterns and emphasize the necessity of upgrading the curves for designing, operating and maintaining municipal water management infrastructures in the future.

ACKNOWLEDGEMENT

This study was co-financed by the Belgian Sci ence Policy Office through the CORDEX.be project

at small spatio-temporal scales. Climatic Change 118: 783–797.

Hassanzadeh, E., Hassanzadeh, A., & Elshorbagy, A. 2014. Quantile-based downscaling of precipitation using genetic programming: application to IDF curves in Saskatoon. Journal of Hydrologic Engineering 19(5): 943–955.

IPCC 2013. Summary for Policymakers. Climate Change 2013: The Physical Science Basis. Contribution of Work ing Group I to the FifthAssessment Report of the Intergov ernmental Panel on Climate Change. In: Stocker,T.F., Qin, D., Plattner, G.-K., Tignor, M., Allen, S.K., Boschung, J., Nauels, A., Xia, Y., Bex, V., Midgley, P.M.e. (Eds.).

Madsen, H., Arnbjerg-Nielsen, K., & Mikkelsen, P.S. 2009. Update of regional intensity-duration-frequency curves in Denmark: Tendency towards increased storm intensi ties. Atmospheric Research 92: 343–349.

Mailhot, A., Duchesne, S., Caya, D., Talbot, G. 2007. Assess ment of future change in intensity-duration-frequency

(IDF) curves for southern Quebec using the Canadian

Regional Climate Model (CRCM). Journal of Hydrology

347: 197–210.

Mearns, L.O., Gutowski, W.J., Jones, R., Leung, L.-Y.,

McGinnis, S., Nunes, A.M.B. & Qian,Y. 2009. A regional

climate change assessment program for North America.

EOS 90(36): 311–312.

Mirhosseini, G., Srivastava, P., & Sharifi, A. 2015. Devel

oping probability-based IDF curves using Kernel den

sity estimator. Journal of Hydrologic Engineering 20(9):

04015002.

Mahoney, K., Alexander, M., Scott, J.D. & Barsugli, J. 2013.

High-resolution downscaled simulations of warm-season

extreme precipitation events in the Colorado Front Range

under past and future climates. Journal of Climate 26:

8671–8689.

Mpelasoka, F.S., & Chiew, F.H.S. 2009. Influence of Rainfall

Scenario Construction Methods on Runoff Projections.

Journal of Hydrometeorology 10: 1168–1183.

Ntegeka, V., Baguis, P., Roulin, E., Willems, P. 2014. Devel

oping tailored climate change scenarios for hydrolog

ical impact assessments. Journal of Hydrology 508C:

307–321. Olsson, J., Berg, P., & Kawamura, A. 2015. Impact of RCM spatial resolution on the reproduction of local, subdaily precipitation. Journal of Hydrometeorology 16: 534–547. Rodríguez, R., Navarro, X., Casas, M. C., Ribalaygua, J., Russo, B., Pouget, L. & Redaño, A. 2014.

Influence of climate change on IDF curves for the metropolitan area of Barcelona (Spain). International Journal of Climatology 34: 643–654. Tabari, H., AghaKouchak, A., & Willems, P. 2014. A perturbation approach for assessing trends in precipitation extremes across Iran. Journal of Hydrology 519: 1420– 1427. Tabari, H., Taye, M.T., & Willems, P. 2015. Water availability change in central Belgium for the late 21th century. Global and Planetary Change 131: 115–123. van der Linden, P., & Mitchell, J.F.B. 2009. ENSEMBLES: climate change and its impacts: summary of research and results from the ENSEMBLES project. Met Office Hadley Centre, Exeter, 160 pp. Wang, D., Hagen, S.C., & Alizad, K. 2013. Climate change impact and uncertainty analysis of extreme rainfall events in the Apalachicola River basin, Florida. Journal of Hydrology 480: 125–135. Willems, P., Arnbjerg-Nielsen, K., Olsson, J. & Nguyen, V.T.V. 2012. Climate change impact assessment on urban rainfall extremes and urban drainage: methods and shortcomings. Atmospheric Research 103: 106–118. Willems, P., 2000. Compound IDF-relationships of extreme precipitation for two seasons and two storm types. Journal of Hydrology 233: 189–205. Willems, P., & M. Vrac, 2011: Statistical precipitation downscaling for small-scale hydrological impact investigations of climate change. Journal of Hydrology 402: 193–205. Willems, P., 2013: Revision of urban drainage design rules after assessment of climate change impacts on precipitation extremes at Uccle, Belgium. Journal of Hydrology 496: 166–177. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Influence of model uncertainty on real-time flood control performance

E. Vermuyten, P. Meert, V. Wolfs & P. Willems

Department of Civil Engineering, KU Leuven, Leuven, Belgium

ABSTRACT: Previous research has shown that intelligent control of hydraulic structures can strongly reduce

flood consequences, in ideal circumstances. However, uncertainties can significantly impact the performance of

real-time flood control strategies. For the Herk river case study in Belgium, this research aims to quantify the

influence of the hydraulic model uncertainty. The flood control is for this case conducted by a combination of

a Reduced Genetic Algorithm (RGA) and Model Predictive Control (MPC) as optimization method. First, the influence of the initial river model conditions and the length of the prediction horizon on the model accuracy are investigated. Next, the performance of the MPC-RGA technique with and without real-time model updating by means of data assimilation is evaluated. Preliminary results show that even a basic data assimilation technique can compensate for some performance loss due to model uncertainty.

1 INTRODUCTION

The historical flood of September 1998 in the Demer basin in Belgium caused a total damage loss of EUR 16 million (HIC 2003). Worldwide, floods are the natural disasters with the highest economic dam age costs. During the last decades, the number of floods strongly increased. The main driving forces of these floods are the increasing trend of extreme rainfall events by climate change (IPCC 2014, Willems et al. 2012, Vansteenkiste et al. 2014) and rising urbaniza tion leading to higher surface runoff and river peak discharges (Poelmans et al. 2011). Because of these ongoing trends, the problem of floods will further expand in the future.

Possible flood hazard reduction strategies are source control measures such as the installation of infiltration facilities and detention tanks at large impervious surfaces, the installation of new retention basins, etc. This paper however focuses on another strategy, namely intelligent real-time control of hydraulic structures. The advantage of this strategy is the optimal use of existing infrastructure and the applicability to densely populated areas lacking space to build new infrastructure.

Model Predictive Control (MPC) is an intelligent control method that uses a process model to predict the future states of controlled variables. With respect to these controlled variables, the optimal set of manipulated variables is determined by means of an optimization algorithm. This method was first used in chemical process industry (Wendt et al. 2002, Nagy 2009), but is nowadays also applied in other domains (Qin et al. 2003) including real-time flood control.

The main problem with the latter application is the nonlinear behavior of the river system during flood events, which requires a nonlinear river model. This turns the problem into a nonlinear programming problem, which is difficult to solve and which can suffer from multiple local optima. Three possible solutions for this problem have been proposed. The first solution is the use of a nonlinear MPC procedure (Barjas Blanco et al. 2010, Schwanenberg et al. 2010). The main obstacles of this method are the computational requirements and global optimality. A second solution is to exclude the nonlinear characteristics from the river model (Breckpot 2013). This method has very long calculation times and was therefore only applied so far for a very small and simple river model. The third solution consists of a combination of Model Predictive Control and a heuristic approach. Because of their ability to deal with nonlinearities, multi-objective analyses and uncertainties, heuristic approaches have gained much interest in recent years (Rani et al. 2010). Successful applications can be found in Van den Zegel et al. (2014), Vermuyten et al. (2014) and Chiang et al. (2015), whereby a Genetic Algorithm (GA) is used in combination with MPC for river flood control. To limit the model computational times, conceptual river models were applied in a way complementary to the detailed full hydrodynamic models. During recent years, the MPC-GA technique has been further advanced by the authors. Larger and more complex river models have been considered, including the floodplains. Fast optimization methods have been implemented that can handle a large number of optimization variables and real economic damage functions as objective to be optimized. For that purpose, the GA was modified into a Reduced Genetic Algorithm (RGA) (Vermuyten et al. 2015). The MPCRGA technique was found to be successful in reducing the flood consequences under ideal circumstances. The next step, which is the focus of this paper, is to take uncertainties into account. More specifically,

this paper investigates the influence of the concep tual model uncertainty on the MPC-RGA based flood control performance.

Section 2 gives a brief introduction to the study area of this research. Next, section 3 describes the used methods, followed by section 4 discussing the results. Section 5 concludes the paper by giving an overview of the main findings of this work and suggestions for future research.

2 STUDY AREA

The study area of this research consists of the Herk river, located in Belgium. The Herk river is part of the Demer basin, which is one of the eleven river basins in Flanders. The basin is very flood prone and has suf fered from many floods (the most recent large floods are the ones in September 1998, January 2002 and November 2010).

The considered part of the basin consists of two streams, the Kleine Herk and the Grote Herk, and eleven floodplains. Three hydraulic structures were installed to regulate the flow. They are at current oper ated individually by Programmable Logic Controllers (PLCs). The primary task of these PLCs is to con trol the inline retention basin and to protect the city of Stevoort, which is located downstream of this retention basin.

3 METHODS

3.1 Model Predictive Control

The current Programmable Logic Controllers are local and instantaneous controllers. This means that each hydraulic structure is regulated individually and con trol actions are only based on the current system states, mainly water levels. Model Predictive Control (MPC), or Receding Horizon Control (RHC), uses a process model to predict future system outputs over a certain time horizon. In this way, optimal future inputs can be determined including the effect of disturbance vari ables on the future system states. This results in a regional, anticipating and dynamic control strategy.

3.2 MPC-RGA

The MPC-RGA technique consists of three building blocks: a river model, a scenario generator and a selec tor (Fig. 1). Each optimization starts with updating the river model by eliminating the systematic difference between model results and observations. This is done by means of data assimilation where the model initial conditions are adapted in real time to bring the model close to the current state conditions. Next, the sce nario generator creates a possible future regulation, a so-called gate level (GL) scenario. Together with rain fall forecasts, this GL scenario is applied to the river model resulting in predictions on the system states over a given time horizon. These states are then compared Figure 1. Scheme of the MPC-RGA technique. with the states of the best regulation so far. The best gate level scenario with respect to the objective criteria is selected and passed on to the scenario generator for the next iteration. This loop is repeated until the stopping criterion is met. Multiple iterations can be executed in parallel due to the specific character of the MPC-RGA approach. During the last step of the optimization, the first gate levels of the best scenario are applied to the river system, whereupon the whole procedure is repeated. 3.2.1 Conceptual river model The river model is a crucial part of the MPC-RGA technique. Since many simulations have to be carried out with this model during optimization, a fast river model is required. Full hydrodynamic models are not suitable for this purpose since they are computationally too slow. Therefore, this research makes use of a conceptual river model. Such a conceptual model is a reservoir-type of model that lumps the hydrodynamic processes in space (storage cell concept). This lumping results in a gain in computational efficiency. The conceptual model in this work was developed with the Conceptual Model Developer tool of Wolfs et al. (2015). This tool applies a modular approach consisting of different model components which can be freely combined, depending on the user needs and available data. The calibration of these components are data-driven, but the obtained parameters have a physical meaning.The calibration data used in this work are simulation results of a detailed InfoWorks RS model (Innovyze 2014) implemented by the Flemish Environment Agency (VMM). The conceptual model consists of multiple reservoirs interconnected by different hydraulic structures. The flows over these structures are calculated by means of the hydraulic structures equations in InfoWorks RS and are based on the upand downstream water level and, if applicable, on the gate position.The water levels in the river system are defined by hypsometric curves. These are (V , h)-relations that transform the reservoir storage to the corresponding water level. At each time step, a continuity equation is applied to calculate the new storage volumes.

3.2.2 Scenario generator

The scenario generator is responsible for creat

ing possible future control strategies. Two different

approaches were implemented.

In the first approach, totally new gate level sce

narios are created. Hereby, random gate positions are

generated over the control horizon with a fixed time

interval. Next, a curve is fitted through the generated

points and the initial gate position. In this way, smooth

GL series are obtained.

In the second approach, new GL scenarios are cre

ated by mutating the best GL scenario so far. There are

two options for this mutating process: diversification

and intensification. At the beginning of both options,

gate levels are selected with a fixed time interval. In

the diversification option, the selected gate position is

set equal to the previous gate level. The intensification option shifts the selected gate position a few centime ters up or down. At the end of both options, a new curve is fitted through the selected gate positions.

3.2.3 Selector

The task of the selector consists of selecting the best control strategy with respect to the objectives. There are four control priorities.

First of all, a safety margin is considered for the water level in the retention basin. Overtopping of this basin could lead to dike failure with huge damage.

Secondly, the total flood damage is minimized.

This damage is computed from the water levels in the floodplains making use of economic damage functions obtained for our case study by the International Marine and Dredging Company. They used the LATIS tool developed by Flanders Hydraulic Research and Ghent University (Kellens et al. 2008). This tools combines land use maps and socio-economic data in order to obtain maximum damage maps. These maps represent the maximum damage cost when the whole area is flooded. Damage factors, depending on the type of land use, are defined to transform these damage maps to the real incurred damage cost as a function of the water depth. By calculating the incurred damage cost for different water levels, a relation between the water level and the damage cost can be defined for each floodplain area. In this research, only industrial and residential damage is considered.

In case there is no flooding or damage, a safety mar gin is considered to keep the river water levels below the crest levels of critical dikes. These critical dikes are defined as dikes for which overtopping would lead to huge damage.

Finally, when all previous criteria would be met, the retention reservoirs are emptied in order to have the storage capacity available for potential new flood events.

3.3 Data assimilation

Data assimilation (DA) is a technique whereby obser vations of the actual system are used to update the model states in real time. Our Model Predictive Controller uses data assimilation at the beginning of each optimization when updating the river model. This feedback mechanism helps to take uncertainties into account. These uncertainties are mainly the model and rainfall forecast uncertainty. In our previous research (Vermuyten et al. 2015), these uncertainties were ignored. This research still assumes the rainfall forecasts to be perfect (without uncertainty), but investigates the influence of the model uncertainty.With this objective in mind, different models are used to represent the actual river system (main model) and to make predictions (prediction model). The data assimilation method used in this work is a very basic one. The method assumes that there are water level observations available in each reservoir of the model. Reservoirs with actual observations available, consider these for updating at the actual locations. For the other reservoirs, the downstream water level location in the reservoir is selected (virtual updating locations). By means of inverse hypsometric curves ((V , h)-relation), the observed water levels are converted to reservoir volumes. This rather simple DA approach has the advantage that is has a low computational cost and that it allows to get a first assessment on the added value of the data assimilation. Additionally, the obtained results can be used as a reference point to study the benefits of installing new measurement devices or more sophisticated DA methods. 4 RESULTS AND DISCUSSION 4.1 Conceptual model Two different conceptual models were set-up in this study, a detailed one (DCM) and a reduced one (RCM). Both models are identified, calibrated and validated based on the simulation results for ten events obtained with the detailed full hydrodynamic InfoWorks RS model (IWRS) of the Flemish Environment Agency. The ten events cover extreme dry as well as extreme wet periods and both artificial and historical events. In the detailed conceptual model, all storage cells of the floodplains are explicitly modeled. In the reduced conceptual model, these storage cells are integrated in the river reservoirs where possible, or merged with each other. This reduction leads to an additional gain in computational time. Figure 2 gives a schematic overview of the reduced conceptual model. The retention basin is covered by reservoirs 2, 9 and 12 (R2, R3 and R12). The most flood damage prone locations are located in reservoirs 10 and 13 (R10 and R13). 4.2 Model accuracy RCM To estimate the loss of accuracy of the reduced conceptual model in comparison with the InfoWorks model and the detailed conceptual model, simulation results of these models were compared. This is done for the same ten events as used for the setup and calibration

Figure 2. Schematic overview of the reduced conceptual

model. Data assimilation updating locations are indicated

with stars and the controllable hydraulic structures with

hol-low boxes.

Figure 3. Model accuracy of the reduced conceptual model

(RCM) for water level predictions at data assimilation

updat-ing locations over the next 48 hours compared to

the In-foWorks model (IWRS) (black) and the detailed

conceptual model (DCM) (grey).

of the conceptual models. The hydraulic structures are in these simulations considered operated by the Programmable Logic Controller. The same PLC reg ulation as in InfoWorks or in the DCM is applied to the RCM as time series. To avoid correlation between different simulations, only every 24 hour a simulation over the next 48 hours is carried out. Each simulation starts with an update of the reduced conceptual model according to the data assimilation approach described in section 3.3. For the analyses of the model accuracy, only simulations with a violation of the critical dike or damage cost criteria are considered. This means the analyses focus on high water periods.

Figure 3 compares the obtained results for the water level at the updating location of each reservoir. The upper graph shows boxplots for the maximum water level (WL) difference and the second one for the dif ference between the maximum water levels. A positive difference indicates an overestimation of the water level by the reduced conceptual model. Figure 3 also

presents the Root Mean Square Error (RMSE) for the Figure 4. Model accuracy of the reduced conceptual model (RCM) for different optimization criteria over a time horizon of 48 hours compared to the InfoWorks model (IWRS) (black) and the detailed conceptual model (DCM) (grey). water level series. A lower RMSE implies a better match. With respect to flood prevention, the results of the maximum water levels are most important. As can be seen in the graphs, the difference of the maximum water level is usually lower than 10 cm. It is clear that the model performs the worst at reservoir R11. This is, however, a less important reservoir, not directly related to flood damage.The most important reservoirs for the real time control are the retention basins R2 and R12 and the damage sensitive locations R9 and R13. These all perform very well. Figure 4 analyzes the accuracy of the reduced conceptual model with respect to different optimization criteria. Graph 4a shows the difference between the predicted and observed total damage cost. Higher differences are always related to higher total damage costs. The MeanAbsolute Percentage Error of the simulations with a damage cost higher than EUR 100 000 is 11%. The average flood duration error over all damage locations is presented in graph 4b. Only simulations with damage are taken into account for both graphs.The critical dike criterion is expressed as a percentage, with 100% indicating an overall margin and 0% overtopping of all critical dikes. Graph 4c displays the error on this factor, excluding errors equal to zero. Finally, graph 4d shows the average error for the maximum retention basin levels. The model shows very reliable results for all optimization criteria. Especially the water levels in the retention basin are very well predicted. In general the accuracy of the reduced conceptual model is very good, bearing in mind the extremity of some events. Typically the accuracy is higher with respect to the detailed conceptual model than to the InfoWorks model. Although total prediction errors are expected to be higher, these model uncertainties were considered useful to study the influence of model

Figure 5. Model accuracy of the reduced conceptual model

(RCM) for different prediction horizons compared to the

In-foWorks model (IWRS).

uncertainty and data assimilation on the MPC-RGA

results.

4.3 Prediction horizon

The prediction horizon is an important parameter of the

Model Predictive Control technique. Long prediction

horizons give the algorithm more time to anticipate

on future rainfall and allow for a more optimal con

trol of retention basins. Rainfall forecast uncertainties, however, strongly increase for longer horizons. A sensitivity study has shown that, for this river sys tem, control horizons longer than 48 hours do not improve the flood control performance when model mismatches and rainfall forecast uncertainties are not taken into account. In order to investigate the influence of the length of the prediction horizon on the model accuracy, Figure 5 compares the control objectives for predictions with the reduced conceptual model in com parison to the InfoWorks model. Each graph displays the empirical cumulative distribution function of the prediction error.

It can be seen from this figure that the reduced conceptual model performs better for shorter hori zons. For prediction horizons longer than 48 hours, the model accuracy remains comparable. Presumably, the rainfall forecast uncertainty will become much more important for longer prediction horizons.

4.4 Initial conditions

The influence of the model updating by data assimila tion is shown in Figure 6. In case of positive updating errors, the reduced conceptual model tends to con sistently overestimate the total damage cost, flood duration and water level in the retention basin and underestimate the critical dike factor. This means the model in general predicts a worse situation than the Figure 6. Model accuracy of the reduced conceptual model (RCM) for different initial value errors compared to the In-foWorks model (IWRS). Figure 7. Sensitivity of the initial conditions in different res-ervoirs on the model accuracy of the reduced conceptual model compared to the InfoWorks model (IWRS). observed one. In case of negative updating errors, the opposite effect is observed. Larger updating errors have a larger effect on the model accuracy. When introducing updating errors for separate reservoirs, the importance of correct initial conditions for each reservoir can be evaluated. Figure 7 shows the 90% quantile of the absolute prediction error for different reservoirs as a function of the updating error. With respect to damage cost, the initial conditions of reservoirs upstream of and at damage locations (R1, R11, R12 and R13) are the most important ones. For the flood duration, only the upstream locations are important (R1, R11 and R12), while for the critical dike factor also the reservoir containing the critical dike locations is important (R3). The initial conditions of the reservoirs representing the retention basin (R2 and R12), and to a lesser extent the reservoirs upstream of

Figure 8. Analysis of the influence of model uncertainty on the MPC-RGA performance, with and without data assimilation.

this basin (R1 and R11), have an important impact on the retention basin water level.

Typically, initial conditions of downstream reser voirs have a small impact on the objectives. Especially overestimations of the initial conditions can have an important impact on the model accuracy. Furthermore, larger updating errors have larger impacts.

4.5 Real-time flood control

To investigate the influence of the model uncertainty on the real-time flood control performance, any of the three models may be considered: the InfoWorks RS model, the detailed and the reduced conceptual model. The reduced conceptual model is the fastest one and therefore has been used as prediction model in this study. The detailed conceptual model is used for rep resenting the actual river system, since interaction with this model can be automated.

Three types of optimizations have been considered. The first one involves optimizations neglecting the model uncertainty. This means the detailed concep tual model is used as prediction and main model. The results from these optimizations are used as a benchmark for other optimizations. The second type of optimizations deals with model uncertainty but without data assimilation. Hereby the reduced con ceptual model is used both as prediction model and main model during the optimization. Afterwards, the obtained MPC-RGA control strategy is applied to the detailed conceptual model. The third type also deals with model uncertainty, but after application of data assimilation as described in section 3.3 to reduce the loss in performance due to the model uncertainty. Fig ure 8 summarizes the results for the three types of optimizations and the six events whereby the PLC reg ulation leads to flood damage. For each optimization type and event, the median of ten optimization results

is considered to assess the variability in the MPC-RGA

results. For the most extreme event, event 5, the three optimizations lead to very similar results, because the whole area is flooded and the influence of the hydraulic structures on the river flows and water levels is limited. Nevertheless, the MPC-RGA technique performs still better for this event than the PLC based regulation. In general, the damage cost is higher when model uncertainty is considered. The basic data assimilation method that is used here, can already reduce the loss in performance due to this uncertainty for most of the events. Damage loss reductions range between 22% and 97%. For event 4, a negative reduction is obtained, which means the data assimilation worsens the performance of the optimization method. The reason for this becomes clear when comparing the optimization results obtained by the reduced and detailed conceptual models.The total damage cost obtained when applying the reduced model is around EUR 185 000, which is much lower than the damage of EUR 300 000 obtained when using the detailed model. When applying the obtained MPC-RGA regulation strategy after use of the reduced conceptual model to the InfoWorks model, a similar damage cost is found as the one obtained after use of the reduced conceptual model. This means that, despite a better performance of the detailed conceptual model for the PLC regulation, this model does not succeed in reducing the damage as much as the InfoWorks model or the reduced model do. The reason for this is the bad performance of the fictive Broad Crested Weir that was introduced in the conceptual model between reservoirs R13 and R4.This weir does not allow to pass as much water as the one in the reduced model and the InfoWorks model, resulting in higher water levels and higher damage costs in reservoirs R13 and R10. After recalibration of this weir, the detailed conceptual model performed more similar as the reduced one and the InfoWorks model. The authors however decided to continue with the original weir implementation, since this has higher model uncertainty, hence allowed to better investigate the influence of such uncertainty. It is clear that the basic data assimilation method used here reduces the performance loss due to model uncertainty if this uncertainty is small. For larger model uncertainties, as is the case for event 4, the data assimilation worsens the performance. It would be useful to further investigate whether more sophisticated data assimilation methods can deal with both small and large model uncertainties. The basic data assimilation method presented here can be used as a reference for such study. Another interesting conclusion is obtained when comparing the optimization results after use of the reduced conceptual model before and after applying these results to the detailed one (second optimization type). The relative order of the optimized regulations for the ten different runs per event is changed with respect to damage cost when applying these to the detailed conceptual model. The magnitude of the different runs are however comparable. This justifies the assumption made by the authors that it is not so important to find the global optimum during the optimization

in MPC-RGA, but that an approximation is sufficient.

Indeed, a better solution in the prediction model leads not always to a better result in the main model due to model uncertainty. It is, however, important that the obtained regulation is close to the global optimum and that the optimization avoids local optima.

5 CONCLUSIONS

Intelligent real-time control of hydraulic structures has proven to be very efficient for flood control under errorand uncertainty-free conditions. How ever, under the presence of model and rainfall forecast uncertainties, the performance of the flood control strategies may be less. This paper analyzed the impact of the uncertainties related to the model simplification from a detailed full hydrodynamic model to a surro gate, reduced complexity model (in order to reduce computational times) on the control performance.

To quantify the uncertainties related to the model simplification, predictions after use of a conceptual model were compared with these after use of the full hydrodynamic InfoWorks RS model or a more detailed conceptual model. Such theoretical study has the advantage that there are no limitations on the avail ability of observations, the model uncertainty can be excluded from other uncertainties and the power of supplementary measurements can be investigated. It is important to notice that the model uncertainty in reality can differ from the one determined here. Nevertheless, the conclusions made are generally valid.

The model accuracy is evaluated with respect to water levels at updating locations and to the optimiza tion criteria of the MPC-RGA algorithm. Prediction horizons smaller than 48 hours reduce model predic tion errors, while longer horizons enlarges these errors. The effect of the latter is however much smaller. Also the initial conditions of the river model have an impor tant impact on the model performance. By introducing updating errors for individual reservoirs, the impor tance of correct initial conditions of each reservoir could be estimated. Especially overestimations of the initial water levels at updating locations tend to result in bad predictions of the optimization criteria.

Finally, a basic data assimilation technique was tested to reduce the loss in control performance due to the model uncertainty. The applied method con sists of an instant update of the model states, based on the current observed water levels assuming one avail able observation per reservoir. This method reduces the loss in performance when the model uncertainty is small. For higher model uncertainties research to more sophisticated data assimilation techniques has to be done. These techniques incorporate for exam ple the quantification of the model uncertainty to improve the model predictions. The data assimila tion method presented in this research can be used as benchmark. Furthermore, the impact of other uncer tainties, especially rainfall forecast uncertainty, has to

Qin, S., Badgwell, T. 2003. A survey of industrial model pre dictive control technology. Control Engineering Practice 11(7): 733–764.

Rani, D., Moreira, M. M. 2010. Simulation-Optimization Modeling: A survey and Potential Application in Reser voir Systems Operation. Water Resources Management 24(6): 1107–1138.

Schwanenberg, D.,Verhoeven, G.,Van Den Boogaard, H., van Overloop, P. J. 2010. Nonlinear model predictive control of flood detention basins in operational flood forecasting. 9th International Conference on Hydroinformatics, Tianjin, China, 7-11 September 2010.

Van den Zegel, B., Vermuyten, E., Wolfs, V., Meert, P. and Willems, P. 2014. Real-time control of floods along the Demer river, Belgium, by means of MPC in combination with GA and a fast conceptual river model. 11th International Conference on Hydroinformatics, New York City, USA, 17–21 August 2014.

Vansteenkiste, T., Tavakoli, T., Ntegeka, V., De Smedt, F., Batelaan, O., Pereira, F., Willems, P. 2014. Intercomparison of hydrological model structures and calibration approaches in climate scenario impact projections. Journal of Hydrology 519: 743–755.

Vermuyten, E., Van den Zegel, B., Wolfs, V., Meert, P. and Willems, P. 2014. Real-time flood control by means of an improved MPC-GA algorithm and a fast conceptual river model for the Demer basin in Belgium. 6th International Conference on Flood Management, Sao Paulo, Brazil, 16– 18 September 2014. Vermuyten, E., Meert, P., Wolfs, V. and Willems P. 2015. Using a fast conceptual river model for floodplain inundation forecasting and real-time flood control a case study in Flanders, Belgium. 21st International Congress on Modelling and Simulation, Broadbeach, Queensland, Australia, 29 November–4 December 2015. Wendt, M., Li, P. and Wozny, G. 2002. Nonlinear chanceconstrained process optimization under uncertainty. Industrial and Engineering Chemistry Research 41(15): 3621–3629. Willems, P., Olsson, J., Arnbjerg-Nielsen, K., Beecham, S., Pathirana, A., Bülow Gregersen, I., Madsen, H., Nguyen, V.-T.-V. 2012. Impacts of Climate Change on Rainfall Extremes and Urban Drainage. IWA Publishing, 252 p., Paperback Print ISBN 9781780401256; Ebook ISBN 9781780401263. Wolfs, V., Meert, P., Willems, P. 2015. Modular conceptual modelling approach and software for river hydraulic simulations. Environmental Modelling & Software 71: 60–77. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Flash flood prediction, case study: Oman

E. Holzbecher, A. Al-Qurashi, F. Maude & M. Paredes-Morales

German University of Technology in Oman, Muscat, Oman

ABSTRACT: In Oman flash floods regularly cause damages on buildings, infrastructure and cost lives. Predic

tion of flood routing can thus be a valuable tool to mitigate the effects of extreme precipitation events. These tools

can be part of warning systems and used for risk maps in urban planning. However, it is a challenging task, due

to severe uncertainties and lack of hydrological parameters as well as the frequent occurrence and importance of

hydrological extremes. While flood routing is general a difficult task in arid and semi-arid climates, the problem

is aggravated in Oman, not only due to geographical conditions, but also due to urbanization etc. Finally we

describe the approach for flood prediction using 2D distributed physically-based integrated modelling.

1 INTRODUCTION

In Oman every year heavy damages are caused by flash floods. Lives and properties are lost, in few, but disastrous events. In Oman such events are aggra vated, due to three reasons: 1. hydrological events are extreme due to climatic conditions; 2. characteristics of watersheds are in favour of flash floods; 3. some regions undergo heavy changes, due to development and urbanization. Special measures for flood predic tion, to be used in urban and regional planning as well as for early warning, are under discussion.

1.1 Hydrological events

Oman is located at the South-Eastern edge of the Arabian Peninsula. The climate is mostly arid, or semi arid, with hot dry summers and mild winters. Annual average rainfall over Oman is 51 mm/a. But rainfall patterns show a high spatial and temporal variability. Despite of this general setting the summer months especially are characterized by extraordinary conditions, when ridges of the Indian Monsoon reach the Omani coast, mainly the Dhofar Governorate at the southern coastal region near Salalah. The characteristic rainfall pattern thus shows high rare peaks, as typical under arid conditions. Peak events alternate with long years of marginal precipitation. In an irregular pattern Oman is affected by cyclones, which cause extraordinary rainfall peaks – and extreme flash floods.

Oman has been hit by many notable floods during its history which caused severe damages to properties and lives. The floods in Oman can be very high, exceeding 10,000 m^3/s (Al Qurashi 2013). Extreme flash floods appear in connection with cyclones. Rainfall events may reach values higher than 400 mm. Most recent notable events were Gonu, which hit Muscat residential area in June 2007 with up to 714 mm/d of rainfall and more than 900 mm in 36 h; and Phet in June 2010 with a maximum of 585 mm/d and up to 603 mm of rain recorded in 38 hours. In June 2015 cyclone Ashooba brought

231 mm rainfall to Masirah island on a single day. 1.2 Watersheds Watersheds in Oman show characteristic features of arid region hydrology. There is no perennial river in the Sultanate. Wadis remain dry for long time periods, separated by runoff induced by heavy rainfall events. Similar to many other regions with rare rainfall events the stream channels in the downstream part are not clearly located. On plain terrain floods may split irregularly into various flow channels, which is difficult to predict. Typical for the given conditions are flash floods, absence of base flow, sparsity of plant cover, high transmission losses, and high amounts of evaporation and evapotranspiration. Typical rainfall-runoff diagrams, measured at two Omani watersheds in the Batinah Governorate are depicted in Figure 1. Concerning the geographical setting Oman shows some features, which are exceptional on the Arabic peninsula. In the North the Hajar mountain strip, running parallel to the coastline, reaches peaks of more than 3000 m above sea level. It separates the populated Al Batinah region from the interior, which is mainly desert. In the wadis heavy rainstorms produce rapid run-off, flowing downstream as flash floods. Al-Rawas & Valeo (2010) analyse twelve Omani watersheds and find drainage, wadi slope, mean elevation and agricultural area as key characteristics affecting mean peak flow. Concerning the flow in watersheds it is important that the Sultanate has put a dam construction program into action in the past decades. Three types of dams

Figure 1. Typical rainfall and run-off diagram, for sta

tions EM503547AF and EB622638AD in wadis Ma’awil and

EL880230AF and EL880143AD in Al Fara.

appear in Oman, characterized by their major purpose:

70 surface retention dams, 32 recharge dams and 12

flood protection dams (MWR 1999, MRMWR 2008).

For flood routing in the downstream region recharge

dams are of highest importance. With a length up to

several kilometers these dams also fulfil a protection

function and serve to prevent progressive saltwater

intrusion.

1.3 Urbanization

With a population of 3.6 million people (2015) on an area of 309500 km 2 Oman is scarcely populated. As most of its area is inhabitable desert, the surplus of an increasing population gathers mainly in few focal points, mainly in the capital Muscat (1.3 mil lion) and Batinah (1 million). The rapid urbanization in these areas poses severe challenges for flood stream routing, as potential flow paths shift due to construc tion of buildings and infrastructure. Such development also has an effect on flood discharge. Soliman (2010) demonstrates the case on wadiAdai and cyclone Gonu, which hit the Omani capital in June 2007.

1.4 Outline of the paper

In 2015 the project entitled ‘Towards a flood-resilient Omani society: improved tools for flood management‘ has started working to investigate the problem of flash floods in Oman. The general objective of the research is to develop improved methods for flood risk man agement. This includes on the one hand improving the scientific knowledge for analysing and assessing flood hazards and flood risks, and on the other hand devel oping a set of decision support tools for flood risk management as well as a public flood information plat form. The project is funded by The Research Council

(TRC) of Oman. We report about the hydrological and hydraulic approaches that are used and under development in the project. 2 HYDROLOGICAL FORECASTING Rainfall-runoff modelling started in 1930s with the introduction of unit hydrographs. In the 1960s continuous, conceptual models were introduced, but with the increased power of computers, physically-based models were introduced in the 1970s. The different methods were mainly developed first for moderate or humid climates, but have been adopted for semi-arid and arid case studies soon after (Wheater et al. 2007, Mujumdar & Nadesh Kumar 2013). Here the description is mainly divided into lumped parameter models on one side and physical based distributed models on the other. The former, based on statistical relations between hydrological variables are in the focus in this section. The latter, derived more or less strictly from basic hydraulic descriptions, are in the focus of the next section. In desert regions ‘localized, intense rainfall of short duration falling on soils with limited vegetative cover generates Hortonian excess overland flow, …which is rapidly transmitted to an ephemeral stream system to generate flood flow. Flood hydrographs commonly have an extremely short rise time (15–30 min), and the floods move downstream as a discrete wave front …over a dry alluvial channel’ (Wheater et al. 1997). First modeling approaches based on such hydrological descriptions. Al-Qurashi (2008) gives an overview on the development of hydrological modelling and forecasting for arid and semi-arid environments. The Mininstry of Water Resources (MWR) initiated hydrological analyses of rainfall and runoff already in the 1990s. The first flood frequency curves for Oman were constructed based on slope area measurements using historical events (MWR 1991). Classification based on rainfall frequency analysis led to three different categories: mountain areas, hill areas and plains areas, for which frequency rainfall intensities were obtained (MRMEWR 2001). For a return period of 100 years and a duration of 1 h the rainfall amounts to 50 mm in plain areas, to 61 mm in hill areas and to 66 mm in mountainous areas. Concerning runoff the correlation analysis revealed that catchment area, wadi length, wadi slope and the percentage of plain areas on the entire watershed area are most important characteristics. Altogether, in Oman the rainfall-runoff relationship shows a high degree of scatter (Al-Qurashi 2008), as also observed in other arid regions. The poor relationship between rainfall and runoff in different watersheds in arid and semi-arid areas can be explained by the spatial and temporal distribution of both rainfall and runoff (Pilgrim et al. 1988). McIntyre et al. (2007) publish a regression analysis on wadi Ahin in Northern Oman. Al-Rawas (2009) deals with the rainfall-runoff characteristics in Omani watersheds, and also discuss the statistical methods under the influence of urbanization. Using regression analysisAl-Rawas & Valeo (2010) show for Oman that urbanization and farming influence discharge peaks. However, they could not confirm a positive correlation between urbanisation and flood peak discharge. They stress the point that the findings for the arid water sheds may not be relevant for the more humid Salalah region in the South. El-Hames includes before men tioned wadi Ahin in his study proposing an empirical method for flood peak predictions in ungauged arid and semi-arid watersheds.

3 HYDRAULIC MODELLING

Hydraulic (or hydrodynamic) modeling of floods extends the hydrological tools in various aspects. The models work with distributed parameters in the model region. Thus it can be combined with techniques that work with parameter distributions, which are avail able nowadays, like remote sensing (Bates 2004). The mathematical algorithms are derived from physical principles. The advantage of the approach is thus that local conditions of whatever kind (hydrodynamic, hydrological, meteorological, geometrical, geographi cal, geological) can be taken into account, in principle.

However, there are drawbacks for such detailed knowl edge: (1) there are increased computational resources required, (2) the increased number of possibly dis tributed parameters requires a sufficient database and pre-processing.

In the process of developing and testing such inte grated models, that contain hydrological and hydraulic features, there are applications focusing on flood fore casting and prediction purposes. Liu et al. (2005) present a distributed model using the TOPKAPI code for flood forecasting in a Chinese watershed. Liu et al. (2003) outline the connection of such models with GIS.

Babister & Barton (2012) distinguish the follow ing model types: 1D, branched 1D, 2D, integrate 1D and 2D and 3D models approaches. A similar distinc tion is made by Sene (2013). In comparison to 1D, 2D models have the advantage that flowpaths are com puted and are not pre-determined. In relation to the full 3D approach they require less storage and execution time. However, they require much more computational resources than simpler models. At the current state of the development of computational technology their requirements can be fulfilled, if the model region is limited.

3.1 Former studies

In an early study Wheater et al. (1997) presented

an integrated model strategy for wadi Ghulaygi in

Oman, combining hydrological modelling with a sim

ple hydraulic construction, i.e. a cascade of one

dimensional elements. Al-Qurashi et al. (2008) and

Al-Qurashi (2008) applied the Kineros and Kineros2 Figure 2. Location of study area and observation points (rainfall and runoff stations) (2016) codes on wadi Ahin in Oman and performed an extensive sensitivity analysis. They found that infiltration rates on the hillslopes, the Manning's roughnesses on hillslopes and channels, and the rainfall to be of most effect on flood peaks – consistent with findings in literature on other arid regions. Saber et al. (2009) develop a spatiotemporal runoff model based on physical processes, including hydrodynamical modelling in the stream channels (1D) and for overland flow (2D). Their model for transmission losses is based on the curve number method (CN), which they apply on three different arid watersheds, one of them in Oman (wadi Al-Khoud). Philipp (2013) presents an integrated system, including hydrological and hydrodynamical components, for flash flood routing under the influence of groundwater recharge dams. The special focus also lies on transmission losses, for which they introduce a resolution of process dynamics for significant losses. The integrated approach is exemplified on the below mentioned wadi Ma'awil (Philipp & Grundmann 2013). 3.2 Model region In the project, mentioned in the introduction, three watersheds were selected for detailed investigation. These are the wadis Ma'awil (835 km 2), Bani Kharus (1183 km 2) and Al Fara, located in the South Batinah region (Figure 2). There are several observation points, at which measurements are performed regularly by the Ministry of Regional Municipalities and Water Resources (MRMWR). In the figure most relevant stations for rainfall and run-off are indicated. The average annual rainfall is 100 mm/a. The decision for these watersheds was taken, because there are sufficient observation points

measuring relevant parameters and variables (1),

because they are representative for many other water sheds in Oman (2), and because flood prediction is an important task (3), as in these catchments flash floods regularly cause drastic damages.

3.3 Data base

In Oman several administrative agencies collect data relevant for flood prediction. The MRMWR is record ing rainfall and runoff, the latter in wadis and in aflaj. Aflaj are a system of water channels, which is tra ditionally used to tap, collect and distribute water, mainly for agriculture (MRMWR 2008), in other parts of the Near East this is known as quanats (also: qanats, Arnon 1992). In entire Oman there are 167 wadi gauge stations to measure wadi flow and to compute flood volumes. The MRMWR also collects other water rel evant material, such as dam and remote sensing data as well as information about groundwater and wells. Urbanization data is gathered by the Ministry of Hous ing, soil type data the from the Ministry ofAgriculture. Digital elevation maps (DEMs) are produced by the National Survey Authority (NSA). As an example Fig ure 3 shows stream channels, dams and elevations for wadi Ma'awil.

3.4 Shallow water equations

For flood routing we utilize the shallow water equa tions (SWE), also known as Saint-Venant equations,

Figure 3. Stream channels, dams and elevation for wadi

Ma'awil watershed that are fundamental for modelling fluid flow in shallow open fluid systems. They are derived from a description using depth-averaged variables Chaudry 2008). The SWE are used for solving problems for example concerning open channels, floods (Bermudez et al. 1991) and other hydraulic phenomena. They can be written as: with total water depth H , water height above reference height η, velocity vector u, acceleration and due to gravity g. In the vector F the contributions of all other forces are gathered.The equations are derived from the volume and momentum conservation principles, formulated using depth-averaged velocity components. The derivation is based on several assumptions: the fluid is incompressible, in the vertical direction there is hydrostatic pressure distribution, depth-averaged values can be used for all properties and variables (except H), the bottom slopes are small, there are no density effects from variable fluid density or fluid viscosity, the eddy viscosity is much larger than molecular viscosity, atmospheric pressure gradient can be ignored, etc. Friction at the walls, i.e. at the interfaces between fluid and solid, is taken into account by an additional term on the right hand side of equation (2), including with Manning coefficient n (Brufau & García-Navarro 2000, Duran 2015). The system of equations (1,2) is nonlinear. 3.5 Software After a thorough examination of different codes for the solution of the SWE the choice was made for ANUGA (Roberts et al. 2015) developed in connection with Australian National University (ANU) and Geoscience Australia. ANUGA is implemented based on the finite volume method, solving the SWE, as presented above. The components of the program are written in the freely available programming language PYTHON (2016). It is mainly operated via the PYTHON language. The connection with a programming language makes the application very flexible, as the user is able to manipulate datasets of various types to be used as input for the program. ANUGA can be used to simulate tsunami, dam break, flood inundations and other types of shallow flows. It can handle wetting and drying processes reasonably (Mungkasi & Roberts 2011) and is able to accurately resolve discontinuous water surface and velocity profiles, such as shock waves or hydraulic jumps. The model region is overlain by a triangular irregular mesh. Thus complex geometries can be handled easily, and sub-regions with higher dynamics can be

dealt with by using local mesh refinement. ANUGA

is equipped with a built-in mesh generator, irregular triangular mesh. Parameter distributions can be converted from a DEM. For example can be converted to points, at which Manning's n coefficient is computed. The results are then imported to ANUGA, where the mesh is automatically created.

The simulation uses time-stepping to proceed with time. Timesteps are usually in the order of seconds only in order to capture the dynamics of the flow. Their size depends on the mesh size and vice versa, according to the Courant-Friedrichs-Levy condition (Shyy 1994). Finer meshes require finer timesteps. In ANUGA timesteps can be calculated automatically depending on the mesh. However, it is also possible to allow changes of the mesh during the simulation. With such adaptive meshing it is possible to refine the mesh only where and when it is needed. This is especially convenient at the front of the flood.

The ANUGA user may choose out of four types of boundary conditions: reflective, transmissive, Dirichlet and time boundary. The reflective condition is to be used at impermeable boundaries, as walls. The transmissive type is used at outlets (discharge boundaries). Using Dirichlet conditions, the model variables have to be specified, which is typically done at inlets. The time boundary is a generalized Dirichlet type conditions,

which allows variable changes with time.
Special operators can be used to consider spe cial hydraulic structures and other circumstances. A culvert operator is included in the software. Other operators have to be defined by the user. In that way initial conditions, land use, groundwater recharge, transmission losses etc. can be included in the model.
Postprocessing is easy in ANUGA using build-in modules. In flood studies it is important to find the highest peak of the flood. One can search for the max imum elevation with a water depth higher than zero. The function can be restricted to a sub-region or a cross-section. Moreover it allows the construction of time series.
Based on a questionnaire concerning 2 approaches, a British report summarizes like this: 'it is recom mended that the benchmarking is undertaken at the ear liest opportunity' (Defra/Environment Agency 2009, 2013). According to the reports benchmarking should be performed on analytical solutions (1), simplified hydraulic processes (2) and published results (3).
Several benchmark studies for ANUGA are pub lished. Mungkasi & Roberts (2012) treat planar and circular dam break problems, water oscillation on a paraboloid channel, and steady flow over a parabolic

obstruction. In a further publication they extend their studies with a dam break involving a dry area, a dam break involving a shock wave, periodic waves on a sloping beach and on a paraboloid channel, (Mungkasi & Roberts 2013). Guerra et al. (2014) demonstrate that the model is capable of computing flows over highly variable topographies, and providing accurate predic tions for wetting and drying processes. An automated

report on all known validation tests is also available (Mungkasi et al. 2015). It contains test against analytical solutions, against reference data and established solutions obtained with other software tools. A comparison of results on dam break problems between the finite volume code ANUGA and the finite element code COMSOL Multiphysics (2016) is currently being performed by the authors (Holzbecher 2016). 4 APPLICATION AND RESULTS From the studied region in Oman wadiAl Fara (see Figure 2) was chosen for a first simulation with ANUGA. Wadi Al Fara is characterized by two upstream tributaries converging at a confluence (Mazahit). In the watershed there is one recharge dam with a capacity of 0.6 10 6 m 3 and a length of 638 m (MRMWR 2008). In the modelling approach, presented here, there is one main channel in the lowlands downstream of the confluence that widens towards the sea. Upstream tributaries and lowland channel were modelled as sketched in Figure 4. The total length of the model is roughly 70 km. Two upstream tributary hydrographs measured on June 4th 2010, when cyclone Phet hit the Omani coast were used as inputs, measured with high resolution (down to 5 minutes) in a storm event. With this approach we predicted flow observed at two downstream locations, one at the confluence the other near to the ocean (and a highway). Simulated hydrographs were compared with observed time series. The comparison is shown in Figure 5, for both input and output time series. Due to the fact that ANUGA requires input for an area, the inflow boundary values of the simulation at Awabi and Shatan tributary differ slightly from the measured values. Moreover, the input was increased by a factor of 1.5 in order to account for the fact that there is runoff from further tributaries, not considered in the model as presented here. We present Figure 4.

Channel-tributary flow scheme for the simulation used in the ANUGA model (left: bird's view, right: 3D); flow from top to bottom.

Figure 5. Results of flash flood routing at different locations

at wadi Fara watershed (black line: measured; coloured lines:

simulated for different Manning coefficients).

results of simulation runs with three different Manning

coefficients: n = 0.01, 0.02 and 0.03.

In the simulation at the confluence there are two main peaks, which can easily be attributed to the peaks of the two tributaries, which appear at different times. The arrival times of the simulated peaks match with the measurements. However, between the two peaks the simulated runoff is too low. This indicates that the additional tributaries, which are not repre sented in detail in the model, may have runoff peaks between those of Awabi and Shatan.

Also near the outlet at the highway peak height and arrival times match with observed values (see bottom sub-figure). Concerning the peak height the transmission losses were the main fitting parameter. Concerning arrival time we observe that the Manning parameter is the crucial parameter. Here the arrival is represented best for n = 0.02.

5 CONCLUSIONS

Flash flood routing and forecasting in arid and semi arid zones in general and in Oman particularly remains a challenging task. The main reason for this is that under the semi-arid conditions floods are generally less predictable, as they are much more determined by meteorological and hydrological extremes. Concerning Oman, its geography with high mountains and

Defra/Environmental Agency 2013. ‚delivering benefits through evidence', Benchmarking the latest generation of 2D hydraulic modelling packages, Report SC120002.

Duran, A. 2015. A robust and well balanced scheme for the 2D Saint-Venant system on unstructured meshes with friction source term, Int. Journal for Numer. Meth. in Fluids (submitted).

El-Hames, A.S., 2012. An empirical method for peak discharge prediction in ungauged arid and semi-arid region catchments based on morphological parameters and SCS curve number, Journal of Hydrology 456–457: 94–100.

Guerra, M., Cienfuegos, R., Escauriza, C., Marche, F., Galaz, C. 2014. Modeling rapid flood propagation over natural terrains using a well-balanced scheme. Journal of Hydraulic Research, 140p. HAL Id: hal-01094954.

Holzbecher, E. 2016. Dam break models for systematic benchmarking of shallow water equation codes (in prepa ration).

Kineros2, 2016. http://www.tucson.ars.ag.gov/kineros/

Liu, Z., Martina, M. L. V., Todini, E. 2005. Flood forecast
ing using a fully distributed model: application of the
TOPKAPI model to the Upper Xixian Catchment. Euro
pean Geosciences Union, Hydrology and Earth System
Sciences Discussions 9(4): 347-364.

Liu, Y.B., Gebremeskel, S., De Smedt, F., Hoffmann, L.,
Pfiter, L. 2003. A diffusive transport approach for flow
routing in GIS-based flood modeling, Journal of Hydrol
ogy 283: 91-106.

McIntyre, N., Al-Qurashi, A., Wheater H. 2007. Regression
analysis of rainfall-runoff data from an arid catchment
in Oman/Analyse par régression de données pluie-débit
d'un bassin aride d'Oman, Hydrological Sciences Journal
52(6): 1103-1118.

MWR 1991. Flood Frequency Curves for Oman, Sultanate
of Oman, Ministry of Water Resources.

MWR 1999. Dams in the Sultanate of Oman, Sultanate of
Oman, Ministry of Water Resources, 83p.

MRMEWR 2001. Design Flood Estimation Guide for Oman,
Sultanate of Oman, Ministry of Regional Municipalities,
Environment and Water Resources (unpublished).

MRMWR 2008. Water Resources in Oman, Sultanate of
Oman, Ministry of Regional Municipalities and Water

Resources, 127p.

Mujumdar, P.P., Nadesh Kumar, D. 2013. Floods in a Chang

ing Climate, Cambridge (UK): Cambridge Univ. Press. Mungkasi, S. & Roberts, S. 2011.A finite volume method for shallow water flows on triangular computational grids, in: International Conference on Advanced Computer Science and Information Systems (ICACSIS), Proc. Mungkasi, S. & Roberts, S. 2012. ANUGA Software for numerical simulations of shallow water flows, Journal of Computer Science and Information 5(1): 1–8. Mungkasi, S. & Roberts, S. 2013. Validation of ANUGA hydraulic model using exact solutions to shallow water wave problems, Journal of Physics: Conf. Series 423, doi:10.1088/1742-6596/423/1/012029 Mungkasi, S., Roberts, S., Davies, G.G., vanDrie, R. 2015. Automated Report on the Performance of ANUGA on (all) Validation Test Problems. 122p. Philipp, A. 2013. Novel Analytical Hydrodynamic Modeling for Evaluating and Optimizing Alluvial Recharge, Doctorate thesis, Dresden: Techn. Univ. Dresden. Philipp, A. & Grundmann, J. 2013. Integrated modeling system for flash flood routing in ephemeral rivers under the influence of groundwater recharge dams, Journal of Hydr. Eng. 139: 1234–1246. Pilgrim, D. H., Chapman, T. G., Doran, D. G. 1988. Problems of rainfallrunoff modelling in arid and semiarid regions, Hydrological Sciences 33(4): 379–400. PYTHON, 2016. https://www.python.org/ Roberts, S., Nielsen, O., Gray, D., Sexton, J., Davies G. 2015. ANUGA User Manual, Release 2.0, Geoscience Australia. Saber, M., Hamaguchi, T., Kojiri, T., Tanaka, K. 2009. Spatiotemporal runoff features of hydrological modeling in Arabian wadi basins through comparative studies, Annuals of Disas. Prev. Res. Inst., Kyoto Univ. 52B: 813–830. Sene, K. 2013. Flash Floods: Forecasting and Warning, Dordrecht: Springer. Shyy, W. 1994. Computational Modeling for Fluid Flow and Interfacial Transport, Dover Publ.: Mineola Soliman, M.M. 2010. Engineering Hydrology of Arid and Semi-Arid Regions, Boca Raton: CRC Press. Wheater, H. S, Woods Ballard, B., Jolly, T.J. 1997. An integrated model of arid zone water resources: evaluation of rainfall-runoff simulation performance, in: Sustainability of Water Resources under Increasing Uncertainty, IAHS Publ, No. 240 (Proc. of the Rabat Symposium): 395–405 Wheater, H.S., Sorooshian, S., Sharma K.D. (eds) 2007. Hydrological Modelling in Arid and Semi-Arid Areas, Cambridge (UK): Cambridge Univ. Press.
Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Equipping the TRENT2D model with a WebGIS infrastructure: A smart

tool for hazard management in mountain regions

N. Zorzi, G. Rosatti & D. Zugliani

Department of Civil, Environmental and Mechanical Engineering, University of Trento, Trento, Italy

A. Rizzi & S. Piffer

Trilogis Srl, Rovereto, Italy

ABSTRACT: Mountain regions are naturally exposed to extreme floods and climate change has worsened

this exposure. Therefore, incisive actions and strategies to safeguard urbanized areas are always more unde

niable. In recent years, reliable mathematical models for hyperconcentrated flows and debris flows have been

developed and used to plan hazard protection and mitigation strategies successfully. However, the increas

ing trustworthiness of advanced modelling implies higher complexity and larger computational burdens,

with a greater request of high-performing hardware. Therefore, new solutions should be found, in order to

overcome these drawbacks and encourage a widespread diffusion of best practices and best available tech

nologies not entailing excessive costs in risk-management field, as the UE Flood Directive (2007/60/EC)

requires. In this work, a smart and easy-to-use solution is proposed and applied to the TRENT2D model,

which is a state-of-the-art 2D model, simulating debris flows and hyperconcentrated flows. This model

was converted into a service and equipped with WebGIS technology, developing a complete, flexible and

user-friendly working environment. This solution allows geographically referenced input and output to be

managed straightforwardly. Moreover, computational burdens are transferred from the user hardware to a

high-performing server, also enhancing model accessibility. To avoid work fragmentation, also a GIS-based

BUWAL-type hazard-mapping procedure was implemented in the same working environment. In this way,

research activity and professional needs are brought significantly closer, encouraging the diffusion of good

practices.

1 INTRODUCTION

In the last years, extreme rainfall events have become more frequent and intense, especially in Europe and in North America (IPCC 2014). In mountain regions, which are naturally exposed to paroxystic events, this trend has caused an increase in the number of extreme floods and geomorphic flows. Moreover, climate change has revealed an increased vulnera bility of urbanised mountain areas, highlighting the necessity of effective protection and mitigation strate gies, able to safeguard population, settlements and infrastructures.

In Europe, guidelines for flood-risk assessment and management are stated by the UE Flood Directive (2007/60/CE), which recommends the application of “appropriate best practice and best available technolo gies not entailing excessive costs” in the field of risk management.

In the last decades, impressive strides have been made in the field of mountain flood modelling, espe cially with regards to hyperconcentrated and debris flows (see Iverson & Ouyang 2015 for an overview), and recent applications have shown that advanced models can support hazard assessment and management effectively (see for instance Rosatti et al. 2015). However, the increasing physical complexity of most state-of-the-art models often implies large computational burdens and long computation time, if standard computers are used. This issue still discourages a wide diffusion of the most cutting-edge models between practitioners and stakeholders, which prefer simpler but untrustworthy approaches and tools. Moreover, advanced models are generally not easy-to-use and require several other stand-alone software to prepare input data and analyse model results. Also complexity and fragmentation of working environments and file formats seem to limit significantly the diffusion of advanced models, hindering an effective management of flood hazard and risk. In this work, a new smart tool supporting mountain hazard management is presented. This infrastructure, called TRENT2D WG, was developed for the purpose of enhancing and simplifying the use of TRENT2D (Armanini et al. 2009, Rosatti and Begnudelli 2013), a state-of-the-art model which simulates hyperconcen trated flows and debris flows. TRENT2D WG is a web solution developed according to the SaaS (Software as a Service) approach, with the model converted into a service and equipped with WebGIS technology. This new infrastructure offers an innovative and complete working environment, where simulations can be per formed easily and input and output geographic data can be processed, organised and displayed straight forwardly. Moreover, also a BUWAL-type hazard mapping procedure (Heinimann et al. 1998), devoted to assess debris-flow hazard, was introduced in the same working environment, taking advantage of the large flexibility of the web infrastructure. In this way, the procedure can be applied directly starting from model results.

The paper is organised as follows. In Section 2 the TRENT2D model is presented. Section 3 is devoted to the innovative technology applied to build the sys tem TRENT2D WG. Then, in Section 4 the system TRENT2D WG, its architecture and its functionali ties are presented, while in Section 5 the integrated BUWAL-type hazard-mapping procedure is described and applied in an educational example.

2 THE TRENT2D MODEL

TRENT2D (Transport in Rapidly Evolutive Natural Torrent) is a 2D model to simulate debris flows and hyperconcentrated flows. Its main properties are shortly described here below (for further details see Armanini et al. 2009 and Rosatti and Begnudelli 2013). Then, data required as input of the model and produced as results are also presented.

2.1 The mathematical and numerical model

TRENT2D is a shallow-flow model, based on a two-phase description of the mixture of water and sediments. No velocity lag is assumed between the solid phase and the liquid phase. The model adopts a mobile-bed approach to represent properly erosion and deposition, which are characteristic processes of the modelled phenomena. Variations of the bed eleva tion are fully-coupled with the mixture dynamics and derive directly from the system of governing equations and from the two-phase approach. The bed elevation z_b is one of the unknowns of the system, just like the flow depth h and the x and y components of the depth-average velocity vector $\vec{u} = (u_x, u_y)$. The values of the unknowns are obtained straightforwardly from the system integration.

Model governing equations describe the conser vation of the mixture mass (Equation 1a), the con servation of the solid mass (Equation 1b) and the conservation of the mixture momentum along the x and y directions (Equations 1c and 1d). Two further rela tions define the relationship between the unknowns,

the concentration c and the shear stresses τ_{bx} and τ_{by}. The system of governing equations can be written as: where c_b is the maximum packing concentration of the bed solid material, c is the depth-averaged concentration, $\delta = 1 + c\Delta$ with $\Delta = (\rho_s - \rho_w)/\rho_w$, where ρ_s is the density of the solid phase and ρ_w is the density of the liquid phase, g is the gravitational acceleration and τ_{bx} and τ_{by} are the x and y components of the bed shear stress $\vec{\tau}$. The concentration c is expressed as a function of the Froude number $Fr = \|\vec{u}\|/\sqrt{gh}$, by means of the Equation 2, originally proposed by Rosatti & Fraccarollo (2006). The non-dimensional parameter β is called transport parameter and depends on sediment shape and diameter. In the model, it is assumed to be constant. A particular approach to estimate β is presented in Rosatti et al. (2015). Equation 3 describes the shear stress τ in the graininertial regime. It was proposed originally by Bagnold (1954) and modified by Takahashi (1978), with $a = 0.32$ (constant). φ is the friction angle of the material of the solid phase, λ is the linear concentration, defined as and Y is the relative submergence, defined as where d is the sediment grain size. The high non-linearity of the governing equations and the presence of non-conservative terms require

a sophisticated numerical model. In this model, the governing equations are solved over a regular Cartesian mesh, by a finite-volume method with Godunov type fluxes. A MUSCL-Hancock approach allows to obtain second order accuracy in space and time. Further details about the numerical model are available in Rosatti & Begnudelli (2013).

The complexity of the mathematical and numerical model leads to high computational burdens, which should be supported by suitable high-performing (and expensive) hardware in order to limit computational time.

2.2 TRENT2D input and output data

TRENT2D simulations can be performed starting from two classes of input data: geographic data and hydrological data. With these data, the model produces geographic data also as results.

The fundamental input geographic datum is represented by the Digital Terrain Model (DTM) of the study area, in ASCII GRID format. DTMs with high resolution are recommended in order to obtain highly reliable results.Then, geographic information could be enriched also by other maps, as for instance orthophotos or thematic layers. All these data can be managed suitably by means of Geographic Information Systems (GISs), which are conceived to display, organise and analyse geographically referenced maps. However, TRENT2D does not support itself the management of geographic data, forcing the user to make the use of stand-alone GIS applications, with consequent work fragmentation.

The other category of input data involves hydrological information, which is essential to compute model boundary conditions. For each inflow section, a hydrograph of the liquid phase is required. Notice that this hydrograph represent the distinguishing characteristic of each scenario to be modelled. Generally, this information is quite simple to obtain in instrumented basins. However, geomorphic flows are observed usually in small non-instrumented basins. Therefore, liquid hydrograph should be obtained applying an appropriate rainfall-runoff model, i.e. starting from rainfall data. If a past event is back-analysed, some measured or interpolated rainfall data could be used, while, in anticipatory study, rainfall data can be estimated from

Instensity-Duration-Frequency (IDF) curves.

Once liquid hydrographs are defined, boundary conditions are computed applying model closure rela tions. Assuming the local uniform flow condition in the upstream section of the study area, it is possible to define suitable a-priori values of the model param eters, which turn out to be reliable when the study area is located far enough from the debris-flow trig gering point (Rosatti et al. 2015). Thereafter, mixture discharge can be computed straightforwardly.

Supplying these data, hyperconcentratedand debris-flow dynamics can be modelled, producing geographic data as results. These maps describe

space and time evolution of the governing variables (i.e. flow depth, deposition depth, erosion depth, velocity, concentration...).Therefore, GIS applications could be used conveniently also to manage and analyse model results. 3 SAAS APPROACH AND WEBGIS TECHNOLOGY The basic ingredients ofTRENT2D WG are two: a particular software-delivering approach, called Software as a Service (SaaS), and the technology of Web-based GIS. Both are presented shortly hereafter. 3.1 The SaaS approach The software-delivering approach SaaS represents a promising alternative to the traditional standalonesoftware logic. According to SaaS, software is developed as a service and hosted by a cloud server. Therefore, applications can be accessed through a suitable Graphic User Interface (GUI) by means a common Internet browser, without limitation on the Operating System and without installation. This approach offers several advantages, in comparison with standalone-software logic. First, the whole computational burden is supported by a suitably equipped server, presumably reducing computational time. Moreover, software can be accessed through the World Wide Web, by a simple login from any Internet-connected device. Furthermore, the service can be used almost independently of device properties. This solution shrinks user burdens significantly, since no high-performing local hardware is needed. In SaaS, multiple users access a single infrastructure, which is centrally maintained and administered. Therefore, all users can access the same software version, making updates available for all the users at the same time and simplifying debug processes. Moreover, this approach brings innovation closer to users, facilitating connections between researchers, specialists, practitioners and stakeholders. Advantages offered by a SaaS solution turn out to be useful also for data storage, since data processed by software can be saved and organized on the same cloud server hosting the service. In this way, the user does not need large local storage capacity. Furthermore, hardware physical damages, virus attacks and blackouts can be excluded, since physical and logical security is ensured by the service provider. 3.2 WebGIS technology The SaaS approach represents the basis also of Webbased GIS (or WebGIS) technology. WebGIS solutions are web applications able to display, organise and process geographic and economic data (Plewe 1997). Clearly, they offer GIS functionalities, which can be accessed through a suitable web GUI.

Figure 1. Main menu of the TRENT2D WG workspace.

WebGIS technology has already been employed in

hazard management, in the form of communication

media (see for instance Hagemeier-Klose & Wag

ner 2009), decision-supporting tool (e.g. Andrienko &

Andrienko 2001) or historic flood database (Miller &

Han 2009). However, the huge potential of WebGIS

technology (De Amicis et al. 2009) has still to be

totally exploited in this field. For example, only few

existing applications combine WebGIS functionalities

and modelling services and most of them were devel

oped for particular and limited applications (see for

instance Kulkarni et al. 2014).

In this work, we try to take advantage of WebGIS technology and its characteristic flexibility, which allows easily to create custom applications, hosting multiple services.

4 TRENT2D WG: AN INTEGRATED SOLUTION FOR HAZARD MODELLING

Complying with the SaaS approach, the TRENT2D model was converted into a service and equipped with WebGIS technology, with the purpose of obtain ing a web smart modelling tool supporting debris and hyperconcentrated-flow hazard assessment. This new integrated solution is called TRENT2D WG and aims to overcome some of the drawbacks typical of advanced modelling. The new solution allows to sim ulate debris flows and hyperconcentrated flow in a user-friendly working environment (Fig. 1), where input data and results can be displayed, organised, overlaid, processed and analysed straightforwardly.

The WebGIS client chosen for the integration is called Terra3 and is presented in Section 4.1. Then, the integration strategy is described in Section 4.2,

together with the architecture of the solution, while Section 4.3 lists some significant functionalities of TRENT2D WG. 4.1 Terra3: a WebGIS application Terra3 is a WebGIS client developed by Trilogis Srl to allow practitioners and stakeholders to deal with geographically referenced data in a user-friendly web environment. It can be accessed from the most common Internet browsers and offers all the advantages typical of a WebGIS solution. Terra3 is built in HTML5, CSS3 and Javascript and employs the interoperable standards of the Open Geospatial Consortium (OGC ©). Thanks to WMS (Web Map Service 1.1.0 and 1.3.0) and WFS (Web Feature Service), it is able to access many geo-data repositories. The application was developed with a modular architecture and was equipped with an intuitive web GUI.The main entrance of the application is a 2D interactive map based on OpenLayer 2.0. Also a 3D view is available, thanks to the Java-based NASA World Wind engine. 4.2 Integration methodology and system architecture The TRENT2D model was exposed as a service on a cloud server hosting the WebGIS Terra3. On the same server also other useful functionalities were exposed, devoted to preand post-process TRENT2D input and output data. Then, all these services were made available and accessible through theWorldWideWeb by means of an intuitive web GUI, developed starting from the Terra3 interface.

Figure 2. Multitier architecture of TRENT2D WG.

The system shows a multi-tier architecture

(Figure 2), where different, but interconnected, soft

ware layers support different tasks. The layers of the

solution are three:

– a presentation tier (or Application Layer), which

allows information to be displayed;

– a logic tier (or Middleware Layer), which is based

on a GeoServer solution and supports services, as

TRENT2D, some modules for data displaying and

other processing functionalities;

– a data tier (or Data Layer), which supports data

storage and retrieving and improves scalability

and performance. A Database Management Sys

tem (DBMS) solution is used to organise geo-data,

which are managed by File System, remaining

independent of server application or business logic.

The infrastructure was developed according to an iterative and incremental approach, which led to a continuous enhancement of the system, with cyclic refinements of the service offering and of the inter face design. In this way, the system was tailored to the requirements of TRENT2D.

4.3 Some functionalities of TRENT2D WG

In addition to the most typical GIS functionalities, TRENT2D WG offers also some other tools, intended to preand post-process modelling data.

First, pre-processing functionalities allow the user to upload and merge DTM ASCII GRID files. Also DTM editing is supported: widespread editing can be performed by means of suitable Shapefiles, while a specific display allows to modify DTM cells point wise.

Other functionalities allow to draw the compu tational domain and to locate the inflow boundary sections, taking advantage of WMS and WFS services.

Once the computational domain is defined, a wiz ard procedure guides the user in the evaluation of the boundary conditions for the model and in the defini tion of the model parameters. In this way, the model can be run easily. In addition, the user can verify simulations progress and analyse partial results while a run is underway. Three different displaying frameworks can be used to analyse model results: – a 2D view, designed for in-depth analysis. Here, results can be analysed point-wise and section and profile charts can be displayed and exported. – A 2D view, introducing model results in a geographically referenced context. This view allows to overview the global dynamics of the phenomena. Time animation is supported. – A 3D view (Figure 3), which operates effectively as communication medium, displaying model runs in a context that is closer to reality, thanks to WMS layers. Furthermore, specific functionalities support download and plot of model results. 5 ASSESSING DEBRIS-FLOW HAZARD WITH TRENT2D WG Hazard assessment can take significant advantage by the use of sophisticated modelling tools. Models allow to reproduce and analyse many different hazard scenarios and represent a precious support in evaluating effectiveness of protection and mitigation measures. Moreover, model results can be used to draw up hazard maps. The TRENT2D model has already been used in debris-flow hazard assessment, with reliable results (e.g., Rosatti et al. 2015). For this reason, the system TRENT2D WG has been enriched with a service devoted to debris-flow hazard mapping. The hazardmapping procedure implemented in the system abides by BUWAL standards (Heinimann et al. 1998) and is based on GIS, as described in Sections 5.1 and 5.2. In Section 5.3, the procedure is applied to an educational case study, showing opportunities offered by the integrated procedure. 5.1 A BUWAL-type hazard-mapping procedure Hazard-level maps, or hazard maps, should be drawn up considering both probability and intensity data, since hazard levels depend on both the occurrence probability of an event and its local intensity. Following the BUWAL approach, the occurrence probability can be taken into account by considering different forcing events, each one characterised by a particular size, i.e. by a significant value of the return period. Commonly, three forcing events, respectively with high, medium and low values of the return period, are considered. On the other hand, local intensity can be defined for each forcing event if some information about the time and space evolution of the phenomenon is provided, namely considering some characteristic physical quantities (e.g. flow depth, velocity, deposition and erosion depths). Comparing local maximum values of these variables with suitable threshold criteria, it is possible to classify the phenomenon local intensity, usually

Figure 3. 3D view in TRENT2D WG.

Table 1. Debris-flow intensity criteria provided by the Autonomous Province of Trento (Italy) with DGP 2759/2006.

Intensity class	Flow depth h (m)		Velocity $\vec{u}$ (ms^{-1})		Deposition depth M (m)		Erosion depth d (m)
High	$h > 1$	or	$\|\vec{v}\| > 1$	or	$M > 1$	or	$d > 2$
Medium	$0.5 < h < 1$	or	$0.5 < \|\vec{v}\| < 1$	or	$0.5 < M < 1$	or	$0.5 < d < 2$
Low	$h < 0.5$	or	$\|\vec{v}\| < 0.5$	or	$M < 0.5$	or	$d < 0.5$

by using three levels (high, medium and low). Generally, intensity criteria are established by national or local authorities. Table 1 shows criteria provided by the Autonomous Province of Trento (Italy) to classify debris-flow intensity.

Then, hazard levels are evaluated by means of a key matrix, called BUWAL matrix, which is a step-wise function depending on probability and intensity.

In this procedure, models turn out to be very useful, because they allow to reproduce time and space evolution of the characteristic variables for each forcing event, i.e. they supply intensity data. Since we are dealing with complex geomorphic flows, only models accounting for physical complexity should be used for hazard-mapping purposes. For instance, models which are not able to represent correctly erosion and deposition processes should be avoided, while models showing high reliability and good forecasting capabilities should be preferred.

5.2 Integration of the hazard-mapping procedure in TRENT2D WG

The model TRENT2D has already been used to assess

debris-flow hazard in several real case studies, with

encouraging results (see for instance Lanni et al. 2015

or Stancanelli and Foti 2015). However, a correct application of the procedure requires a lot of geographically referenced maps, which make hazard mapping a long and laborious task. This happens also if a GIS-based environment is used to organize, process and overlay intermediate maps drawn up by the procedure and to produce and display final hazard maps. For these reasons, the procedure was automated and integrated in the systemTRENT2D WG as a service. The service is called Hazard Mapper and hosted by the Middleware Layer. In this way, the procedure is made available in the same working environment where TRENT2D can be applied and where GIS functionalities are offered. By means of the Hazard Mapper, hazard maps can be produced easily, starting from results supplied by TRENT2D. The service classifies intensity, probability and hazard levels automatically, ensuring a right implementation of the procedure and drawing up readable hazard maps. In this way, the hazard-mapping task is made extremely straightforward and its reliability is preserved. 5.3 An educational application: assessing debris-flow hazard in different design scenarios The systemTRENT2D WG was created as a modelling tool suitable for different purposes. For example, it can

Figure 4. Comparison between the hazard map obtained for the basic configuration (on the left) and the hazard map obtained

for the design configuration (on the right). High hazard level areas is represented in red, medium level in blue and low level

in yellow.

support profitably the design of effective protection

and mitigation measures. In this case, models can be

applied to evaluate the efficiency of such structures, assess their effect on different hazard scenarios or compare multiple design solutions, quantifying hazard variations.

In this work, TRENT2D WG was applied to an edu cational case study, aiming to evaluate the impact of a designed slit-check dam on debris-flow hazard lev els in an urbanised mountain area. For this purpose, a realistic alluvial fan was considered and two differ ent hypothetical configurations compared. In the first configuration, the geographic datum was represented by the original topography of the fan, without any pro tection structure. In the second configuration, a design solution was introduced, with a slit-check dam located at the fan apex. The dam was supposed to control a suitably designed deposition area, containing a maxi mum volume of sediments of about 4000 m^3. The slit was assumed totally obstructed by debris during the simulated events. All these design characteristics were introduced in the system TRENT2D WG thanks to the DTM-editing functionalities, which allow to modify cell elevations easily.

The Hazard Mapper service was used to produce a hazard map for each configuration, for the purpose of quantifying the effect of the dam on hazard levels on the alluvial fan.

Scenarios required by the hazard-mapping proce

dure were simulated by means of TRENT2D, consid

ering the same a-priori values of the model parameters

and the same boundary conditions for both the config

urations. Each scenario was characterised by proper

forcing events, i.e. by boundary conditions obtained

from liquid hydrographs representative of the required

return periods. Liquid hydrographs were generated

starting from probabilistic observations on rainfall, i.e. from suitable Intensity-Duration-Frequency curves, and applying the rainfall-runoff model Peakflow (Rigon et al. 2011). Rainfall return periods of 30, 100 and 200 years were considered. Then, TRENT2D outputs were processed through the Hazard Mapper, which produced the maps shown and compared in Figure 4. On the whole, both the maps cover approximately the same area, with the map of the basic configuration which is slightly wider. On the contrary, the extension of each hazard level changes between the first configuration and the second. Areas with high hazard level, i.e. red areas, are quite wider in the basic configuration, especially in the upper part of the figure. In the second configuration, some of these areas are classified as medium hazard level areas (in blue). Similarly, some of the blue areas of the basic configuration change into low hazard areas (in yellow) in the design configuration. This is a clear proof of the effect of the design structure, which reduces hazard levels on the alluvial fan, especially in its lower part. This result accounts for the deposition processes induced by the slit check-dam in the deposition area, which reduces the amount of solid material reaching the fan. Such analyses were performed quite easily by means of TRENT2D WG, taking advantage of the integration of different tools in a unique and smart environment. Of course, also other design solution could be considered and compared, with different results. 6 CONCLUSIONS In this work, the new modelling infrastructure TRENT2D WG was presented. It was developed as a smart web system, able to overcome most of the issues

related to the use of the TRENT2D model. TRENT2D

WG allows to simulate debris flows and hypercon centrated flows, applying a state-of-the-art model in a user-friendly environment. The system offers sev eral preand post-processing functionalities and most of them GIS-based. Moreover, model results can be used straightforwardly to assess hazard levels, produce hazard maps and evaluate different hazard scenarios, as shown in the educational application.

Joining SaaS approach and WebGIS technology, this system is intended to introduce significant opera tive advantages for the user, encouraging the diffusion of state-of-the-art models between practitioners and stakeholders and bringing research targets closer to professional needs.

In the future, the characteristic SaaS flexibility will allow also other models, e.g. rainfall-runoff or avalanche models, to be integrated in the same sys tem, developing a more complete and versatile work ing environment for hazard management in mountain regions.

ACKNOWLEDGEMENTS

This work was supported and funded by CARITRO Fundation – Cassa di Risparmio di Trento e Rovereto (Italy). Moreover, the authors would like to thank Laura Rech for her precious support during the TRENT2D WG testing stage.

Andrienko, N. & G. Andrienko (2001). Intelligent Support for Geographic Data Analysis and Decision Making in the Web. Journal of Geographic Information and Decision Anaysis 5(2), 115–128.

Armanini, A., L. Fraccarollo, & G. Rosatti (2009). Two dimensional simulation of debris flows in erodible chan nels. Computers & Geosciences 35(5), 993–1006.

Bagnold, R. A. (1954). Experiments on a Gravity-Free Dis persion of Large Solid Spheres in a Newtonian Fluid under Shear. Proceedings of the Royal Society A: Mathematical, Physical and Engineering Sciences 225(1160), 49–63.

De Amicis, R., G. Conti, S. Piffer, & B. Simões (2009). Open challenges and protection of the environment, pp. 265–286. Springer Netherlands.

Hagemeier-Klose, M. & K. Wagner (2009). Evaluation of flood hazard maps in print and web mapping services as information tools in flood risk communication. Natural

Hazards and Earth System Science 9(2), 563–574. Heinimann, H. R., K. Hollenstein, H. Kienholz, B. Krummenacher, & P. Mani (1998). Methoden zur Analyse und Bewertung von Naturgefahren. Technical Report 85, Bundesamt für Umwelt, Bern. IPCC (2014). Climate Change 2014: Sysnthesis Report. Contribution of Working Groups I, II and III to the Fifth Assessment Report of the Intergovenmental Panel on Climate Change. Geneva, Switzerland: IPCC. Iverson, R. M. & C. Ouyang (2015). Entrainment of bed material by Earth-surface mass flows: Review and reformulation of depth-integrated theory. Reviews of Geophysics 53(1), 27–58. Kulkarni, a. T., J. Mohanty, T. I. Eldho, E. P. Rao, & B. K. Mohan (2014). A web GIS based integrated flood assessment modeling tool for coastal urban watersheds. Computers and Geosciences 64, 7–14. Lanni, C., B. Mazzorana, P. Macconi,

& R. Bertagnolli (2015). Suitability of monoand two-phase modelling of debris flows for the assessment of granular debris flow hazards: insights from a case study. In G. Lollino (Ed.), Engineering Geology for Society andTerritory -Volume 2, Acts of: InternationalAssociation of Engineering Geology and the Environment IAEG XII Congress, pp. 1629–1633. Springer International Publishing. Miller, H. J. & J. Han (2009). Geographic data mining and knowledge discovery. Bristol: CRC Press. Plewe, B. (1997). GIS-Online: Information Retrieval, Mapping and the Internet. Santa Fe: OnWord Press. Rigon, R., P. D'Odorico, & G. Bertoldi (2011). The geomorphic structure of the runoff peak. Hydrology and Earth System Sciences 15, 1853–1863. Rosatti, G. & L. Begnudelli (2013). Two-dimensional simulation of debris flows over mobile bed: enhancing the TRENT2D model by using a well-balanced Generalized Roe-type solver. Computers & Fluids 71, 179–195. Rosatti, G. & L. Fraccarollo (2006). A well-balanced approach for flows over mobile-bed with high sedimenttransport. Journal of Computational Physics 220(1), 312–338. Rosatti, G., N. Zorzi, L. Begnudelli, & A. Armanini (2015). Evaluation of the Trent2D model capabilities to reproduce and forecast debris-flow deposition patterns through a back analysis of a real event. In G. Lollino (Ed.), Engineering Geology for Society and Territory – Volume 2, Acts of: InternationalAssociation of Engineering Geology and the Environment IAEG XII Congress, pp. 1629–1633. Springer International Publishing. Stancanelli, L. & E. Foti (2015).A comparative assessment of two different debris flow propagation approaches – blind simulations on a real debris flow event. Natural Hazards and Earth System Sciences 15, 735–746. Takahashi, T. (1978). Mechanical characteristics of debris flow. Journal of the Hydraulics Division 104(HY8), 1153–1169. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Reservoir operation applying a discrete hedging rule with ensemble

streamflow prediction to cope with droughts

S. Lee & Y. Jin

Civil Engineering, Pukyong National University, Busan, Republic of Korea

ABSTRACT: Hedging operation of reservoirs may be a useful method of water supply for drought periods. A

reservoir operation method was studied, which included a derivation of discrete hedging rule applying a linear programming, an ensemble streamflow prediction, and reservoir simulations observing the derived hedging rule comprising four phases. Dam operators supply water based on the available water and the phase in which the available water is. Available water of a reservoir is defined by the current storage plus the inflow volume of the future time period. The future inflow volume was computed by ensemble streamflow prediction using historical weather condition scenarios and precipitation forecast. Reservoir operations of Hapcheon Dam, Republic of Korea were simulated applying the derived hedging rule curves and ensemble streamflow predictions. The results showed the reduction of maximum and overall water supply deficit. The derived hedging rule curves are certainly worth practical reservoir operation for water supply.

1 INTRODUCTION

Water supply deficit may be occurred in drought peri ods and it induces damage to societies. Reservoirs can be operated to reduce the damage applying some hedging rules. Shih and Revelle (1994) developed a continuous hedging rule that comprised two hedging phases. Hedging rule allows releases to be curtailed below target levels to avoid potentially more severe shortages later (U.S. Army Corps of Engineers, 1996). They suggested a discrete hedging rule that stem from a zone-based method. Neelakatan and Pundarikanthan

(1999) and Kim et al. (2014) applied a discrete hedging rule to reservoir systems.

Water supply from the discrete hedging rule sug gested by Shih and Revelle (1995) is based on the available water and the phase in which the available water is.The available water of a reservoir is defined by the current storage plus the inflow volume of the future time period. Although a method to estimate inflow volume of the future time period was beyond the con tents of the research of Shih and Revelle (1995), it is essential to practical reservoir operation. The ensem ble streamflow prediction is a typical method that estimates future inflow volume to a reservoir.

The purposes of the research include a deriva tion of discrete hedging rule comprising four hedging phases and reservoir simulations applying an ensem ble streamflow prediction to estimate future inflow volume, observing the derived hedging rule.

2 RESEARCH MEHTODS

2.1 Discrete hedging rule

U.S.Army Corps of Engineers (1996) describes a vari ety of reservoir operation rules: standard operation rule, hedging rule, pack rule, and zone-based operation rule. The hedging rule can be an effective method for water supply to cope with droughts. The hedging rule rations releases below target levels and can lessen future shortages. Shih and Revelle (1995) presented a discrete hedging rule that uses the trigger volume to ration the release of each hedging phase (Figure 1). Optimization model of the discrete hedging rule decides the trigger volumes, V_{1p}, V_{2p}, and V_{3p}, for the different rationing phases in every months p. Their optimization model maximize the number of releases meeting the full demand for the whole periods and make trigger volumes as small as possible. The objective function (1) is as follows: where $y_{1t} = 1$ if full demand is available during a period t or 0, otherwise; ω= a weight value; V_{1p} = the value of storage and inflow above which no restrictions on water use are placed; V_{2p} = the value of storage and inflow below which phase two is implemented for month p; and V_{3p} = the lower bound of storage plus inflow for month p (Shih and Revelle, 1995). 2.2 Ensemble streamflow prediction The discrete hedging rule described in the previous section is based on the available water. The available water of a reservoir is defined by the current storage plus the inflow volume of the future time period. The future inflow volume was computed by ensemble streamflow prediction (ESP) using a watershed runoff model, historical weather condition scenarios, and precipitation forecast. ESP requires a method assigning

Figure 1. Schematic diagram of discrete hedging rule (Shih and Revelle, 1995).

a weight value to each streamflow scenario. A typical decision method of the weight value is Croley method. The weight values by Croley method are decided by a nonlinear programming that has the objective function (2) and constraints (3).

where w_i = the weight value that is given to i histori cal scenario; e_k = the probability that the precipitation forecast is in the k-th precipitation interval; and $a_{k,i} = 1$ if a historical precipitation scenario value lies in the k-th precipitation interval or 0, otherwise.

2.3 Reservoir simulation

A schematic diagram of reservoir simulation cop ing with droughts is shown in Figure 2, which applies the discrete hedging rule and ensemble stream flow prediction. The reservoir simulation procedures include deciding trigger volumes to ration water sup ply, ensemble streamflow prediction for inflow of the future time period, and reservoir simulation using the hedging rule and the predicted inflow.

3 RESULTS AND ANAYSIS

3.1 The study area

The optimization model deriving the discrete hedging rule was applied to Hapcheon Dam (Fig. 3). Hapcheon Dam is a multipurpose dam that was constructed in 1988 and is located on the upstream region of the Hwang River in Republic of Korea. The project data of Hapcheon Dam are as follows: watershed area is 925 km^2, effective storage capacity is 0.56 billion m^3, and the installed capacity of power generation is 101 MW.

The inflow of Hapcheon Dam has been recorded

since 1990. There were three drought periods in that Figure 2. Schematic diagram of reservoir simulation applying discrete hedging rule with ensemble streamflow prediction. Figure 3. Hapcheon Dam basin. area after the construction as follows: 1994–1996, 2000–2001, and 2008–2009. 3.2 Discrete hedging rule of four phases The Korean drought emergency plan (Ministry of Land, Infrastructure and Transport, 2015) comprises four stages: concern, caution, alert, and severe. The rationing phases of the discrete hedging rule need to be matched with the above four stages. The four rationing phases and their trigger volumes used in the research are shown in Figure 4. Two decision variables of trigger volumes are added to equation (1) and the objective function (4) is as follows:

Figure 4. Schematic diagram of discrete hedging rule with

four rationing phases.

The optimization model has twenty-five constraint

equations. Typical and major constraints are written

here and one can refer to Jin (2016) for the other

constraint equations.

Constraint set (5) in combination with integer

requirements says that if the storage at the end of last

period, S_{t-1}, plus projected inflow, $\hat{I}_t$, is greater than

or equal to the trigger volume V_{1p}, then y_{1t} will be one;

i.e. that no rationing is needed. Constraint (6) in com

bination with the integer requirements requires that y_{1t}

be zero if available water is less than V_{1p}. If there were

no ε in constraint (5), when

the variable y_{1t} could be zero or one. To correct this,

ε is incorporated in to constraint (5) so that when

the variable y_{1t} must be equal to 1.

The variables of y_{2t}, y_{3t}, and y_{4t} have similar pair of

constraint equations to (5) and (6), respectively.

The variables, R_t, means the release at a time period

t. Constraint set (7) set release equal to full demand if

y_{1t}, y_{2t}, y_{3t}, and y_{4t} are unity. Constraints (7) further set release to $\alpha_{4p} D_p$ if y_{1t}, y_{2t}, y_{3t}, and y_{4t} are not one. In

this model, the fraction α_{4p} of water demand is always

released.

where α_{1p}, α_{2p}, α_{3p}, and α_{4p} = percentage of demand

that obtains during phase one, two, three, and four Figure 5. Derived hedging rule curves of Hapcheon Dam. Figure 6. Comparison of water supply from 1992 to 1993. rationing that is different every month; D_p = target release that is different every month; and p = month. Constraint set (8) is the mass balance equation of inflow and outflow in each time period t. There are many ways to express this mass balance. It is assumed that no significant evaporation or seepage losses occur. The mass balance equation states that the storage at the end of the current period is equal to the storage at the end of the previous time period plus any inflow minus the release and any spill in this period (Shih and Revelle, 1995). where W_t = spill in period t. The optimization model described above was solved by a mixed integer programming and the model was applied to Hapcheon Dam. Figure 5 shows the hedging rule curves drawn by the trigger volumes derived from the optimization model. The Box-Whisker plots in the figure were drawn by the monthly available water volume of historical data. Figures 6 to 8 compare the historical release data and the releases of the optimization results for drought periods. It can be seen from the results that the hedging rule operation diminishes water supply shortages and prevents additional releases for hydropower generation that exceeds the water demand.

Figure 7. Comparison of water supply from 1994 to 1997.

Figure 8. Comparison of water supply from 2000 to 2002.

Figure 9. Runoff hydrographs produced by calibration and

verification of tank model.

3.3 Ensemble streamflow prediction

3.3.1 Calibration and verification of watershed runoff model

The tank model with soil moisture structure was used

to simulate historical ensemble streamflow scenar ios. The parameters of this model were estimated

by applying Shuffled Complex Evolution-University

of Arizona algorithm (Duan, 1991). The calibration

period is from 1990 to 2003. The model were verified

for the period from 2004 to 2005. The longer the ver

ification period is, the worse the model performance

become. The reservoir simulation studied here, how

ever, used the dam inflow of the next month and the

performance for a short period of one month was good

and enough to the reservoir simulation. Figure 9 shows

the results of the calibration and verification. Table 1

shows the evaluation results by NSE (Nash-Sutcliffe Table 1. Calibration and verification results for tank model. Simulation PBIAS Item period NSE (%) PEE ROV Calibration 1990 ~ 0.877 1.051 0.951 0.980 2003 Verification 2004 ~ 0.898 −6.616 2.000 1.066 2005 Figure 10. Comparison of Hapcheon Dam inflow predictions. Table 2. Evaluation on Hapcheon Dam inflow predictions. Uniform Croley Index weight value weight value RMSE 30.5 30.5 PBIAS (%) 17.2 6.1 efficiency), PBIAS (percent bias), PEE (proportional error of estimate), and ROV (ratio of volume). 3.3.2 Results of ensemble streamflow prediction To verify the practical applicability of the derived hedging rule, the following reservoir simulation incorporated ensemble streamflow prediction. The ensemble streamflow prediction was performed in two steps: watershed runoffs for each month were simulated using the historical precipitation and potential evapotranspiration under the soil moisture state at the beginning of reservoir operation; an ensemble streamflow prediction value was computed by a weighted runoff using precipitation forecast and weighting by Croley (2003) method. Figure 10 and Table 2 show the comparison results of the ensemble streamflow prediction applying Croley and uniform weighting methods. Croley weighting method yielded better results than those from uniform weight in PBIAS. 3.4 Results of reservoir simulation Reservoir simulation were performed applying the derived hedging rule curves and the ensemble streamflow predictions from 2006 to 2010.The water supplies

Figure 11. Water supply resulted from the discrete hedging rule and ensemble streamflow prediction.

Table 3. Critical values of monthly water supply.

Item	Observed	Simulated
Minimum water supply (10^6 m^3/month)	8.17	20.79
Maximum deficit for the minimum water supply (10^6 m^3/month)	46.20	34.16
Occurrence	July 2009	June 2009

from the reservoir simulation are shown in Figure 11. The hedging operation of the reservoir hindered release for hydropower generation that exceeded the target demand. It also diminished water supply deficit. Table 3 shows specific values of water supply. The minimum water supply resulted from the reservoir simulation is 2.7 times bigger than that of the histor ical record. The maximum water supply deficit from the reservoir simulation is 26% smaller than that of the historical record.

4 CONCLUSIONS

Droughts are recurrent atmospheric phenomena and require a proper reservoir operation method for water supply considering droughts. Hedging rule curves were derived for Hapcheon Dam, Republic of Korea to cope with droughts by an optimization method that

was solved by a mixed integer programming. Practical

reservoir operation also needs a prediction method for

future inflow to a reservoir. An ensemble streamflow

prediction method was applied to estimate the inflow

Determination of the interaction between surface flow and drainage discharge

S. Kemper, A. Schlenkhoff & J. Balmes

University of Wuppertal, Wuppertal, Germany

ABSTRACT: Managing urban flooding caused by heavy rainfall events requires new design approaches con

cerning the drainage system as well as the temporary surface water runoff. Latest developments on bidirectional

coupled models – considering the surface water runoff and the sewer discharge – are still employed in practice.

Connecting elements between the surface and the underground drainage system are the so called street inlets

or gullies – offered in different construction types and designs. Depending on the longitudinal and transver

sal slope of the street as well as the street inlet type the hydraulic efficiency of grate inlets (typically used in

Germany) is hardly available, thus, physical model test runs were done to receive the requested information. Due

to steep longitudinal slopes up to 10%, only supercritical flow conditions occur with flow depths up to 3 cm and

flow velocities of approximately 1–2 m/s. The intercepted flow varies between $Q_I = 2.95$ l/s and $Q_I = 16.00$ l/s

within the investigated model test runs. With three-dimensional numerical model test runs, the capacity of the

underground system (sump with sludge bucket and connection pipe) to cope with the intercepted surface flow is analyzed. Several conditions – clogging effects as well as different pressure heads in the pipe system – were investigated. Based on the results from the numerical model test runs, a chart was developed to identify the maximal possible inflow from the surface to the underground system.

1 INTRODUCTION

According to an increase in the number and inten sity of heavy rainfall events, potentially caused by the climate change (IPCC 2014), managing urban flooding becomes more important in the future. After Digman et al. (2014) four key parts to manage surface water exist (four domains approach). Rainfall events are divided into (1) everyday rainfall, (2) drainage design rainfall, (3) exceedance rainfall and (4) extreme rainfall. Typically, drainage systems are designed for domains one and two. Classical urban drainage sys tems e.g. in Europe mainly consist of underground infrastructures. The surface runoff is captured by so called street inlets while the inlet is connected to the underground piped drainage system by a connection pipe (Fig. 1). According to DIN 4052-2 (2006) street inlets consist of a grate located at the same level as the road and of a sump including a sludge bucket under neath. Regarding Digman et al. (2014) urban areas need to be designed to cope with flooding from the drainage system and manage the exceedance that occur when the drainage system can no longer cope (domain three). In case of reaching domain four (extreme rain fall) emergency plans must exist. In the present paper the exceedance rainfall conditions will be considered.

Managing urban flooding caused by exceedance rainfall events requires new design approaches con cerning the underground drainage system (minor sys tem) as well as the surface runoff using topography Figure 1. Street inlet with connection pipe (DIN 4052-2 2006), modified. and streets (major system) (Fratini et al. 2012). Latest developments on bidirectional coupled models, 1D1D as well as 1D-2D models, are still employed in practice. After Butler & Davies (2011) these models are capable of representing the interaction of the major and minor systems under extreme flow conditions.

Figure 2. Street inlet types (Brown et al. 2009), modified.

1D-1D models (one-dimensional pipe flow model and one-dimensional surface flow model) give sufficient results as long as the surface flow stays within the road cross-section. If the surface flow exceeds the capacity of the street profile, 2D surface flow models pro vide likely more realistic results since inundation areas with corresponding flow depths and flow velocities were calculated. As mentioned, connecting elements between the surface and the underground drainage sys tem are street inlets. There are different existing types, e. g.: (a) grate inlets, (b) curb-opening inlets (c) combi nations inlets and (d) slotted drain inlets (Brown et al. (2009), Fig. 2).

In previous physical model test runs, the grate capacity of selected grate inlets – neglecting the connected bucket and sump underneath – was mea sured (Kemper & Schlenkhoff 2015). The aim of the present paper is to investigate the ability of the whole "street inlet and pipe" system to cope with the intercepted flows under certain conditions using a three-dimensional numerical model.

2 MODEL TESTS

2.1 Physical model: grate capacity

The grate capacity measurements were done in a flume made of acrylic glass with L_{Flume} = 10.0 m in length and W_{Flume} = 1.5 m in width. The bottom roughness is approximately k = 1.5 mm (roofing paper). The slope is adjustable in longitudinal and transverse direction. In an opening area of 500 mm × 500 mm the grate of street inlets can be integrated (scale 1:1). All of the presented investigations and results were done with a standardized grate inlet described in DIN 19583-2 (2012) (Figure 3).

In German guidelines, rainfall intensities with a return interval of 2 to 5 years and a connected catch ment area of 400 m^2 are recommended for the design of street inlets. That implies a surface runoff approaching Figure 3. Physical Model. Table 1. Surface runoff approaching at one inlet. T [a] 0.5 1.0 2.0 5.0 10.0 20.0 50.0 100.0 D [min] Q [l/s] 5 4.3 6.8 9.3 12.6 15.1 17.6 20.9 23.4 10 3.8 5.5 7.1 9.2 10.8 12.4 14.6 16.2 15 3.3 4.6 5.8 7.5 8.7 10.0 11.6 12.9 Table 2. Investigation program: physical model. Longitudinal Slope S L [%] 2.5, 5.0, 7.5, 10.0 Transverse Slope S T [%] 2.5 Discharge Q [l/s] 3, 6, 9, 12, 15, 18, 21 at one street inlet of approximately Q = 9 l/s for a duration of 10 minutes (rain event with a return interval of 5 years located in Wuppertal, a city in the western part of Germany; KOSTRA-DWD-2000 (2000), Table 1). Considering exceedance rainfall events within the physical model tests, the surface runoff was varied between Q = 3 l/s up to Q = 21 l/s with ΔQ = 3 l/s (Table 2). Flow depths were measured upstream of the inlet with three ultrasonic sensors. Using platform load cells the volume of the water intercepted by the grate, the water flowing over the grate inlet and the water flowing beside the grate inlet was measured over the time. Steady flow conditions with a nearly uniform flow were reached upstream of the inlet. For all investigated discharges and slopes, supercritical flow conditions occur, thus, the outflow at the lower boundary of the model was a free outflow with no backwater effects – the same for the outflow through the grate inlet. 2.2 Numerical model: street inlet-system The CFD software FLOW-3D v.11.1 (Flow Science Inc.) was used for the numerical simulations. The

Figure 4. Numerical model.

model geometry corresponds to the requirements

given in DIN 4052-2 (2006). Model and prototype

scale are the same. Therefore no scaling effects occur.

The sludge bucket has a diameter of 300 mm with

4 rows of slots and nine openings in the bottom;

the inner diameter of the connection pipe is 150 mm;

the inner diameter of the sump made of concrete

is 450 mm. The sheet thickness of the bucket was

increased from originally t = 1.25 mm to t = 6 mm to model the geometric regions precisely using the FAVOR TM method (Fractional Area-Volume Obstacle Representation, Flow Science Inc. (2015)) within the rectangular grid. Furthermore, the conical geometry of the bucket was idealized with cylindrical geome try. Due to the small slots in the sludge bucket, the mesh size was set to dx = dy = dz = 4 mm. Approxi mately 10 Mio. cells were used. The RNG turbulence model was used (Renormalized group, based on a k-ε turbulence model).

One aim of the present investigation is to calculate the capacity of the sump with sludge bucket and con nection pipe to cope with the intercepted grate-flows in different conditions – an empty bucket as well as a partly filled bucket up to a fully filled bucket with leaves and sludge. Within the test runs C0, the capac ity with an empty sludge bucket and several inflows MR1–MR10 defined by a source located close to the bottom of the bucket was investigated. The location of the source was selected in order to avoid splashing water from the top.The connection pipe outflow is con trolled by a pressure boundary condition to consider the pressure head in the underground drainage system. Three different pressure heads were defined (P1 – P3, see figure 4). Afterwards, simulation runs with differ ent amounts of clogged slots in the bucket (C1 – C4)

were done to analyze the influence of clogging.

3 RESULTS AND DISCUSSION

3.1 Grate inlet capacity

From the physical model test runs a dependency curve

with intercepted flow depending on the approaching Table 3. Investigation program: numerical model. Effect of pressure head h p MR 1 Q = 10 l/s P1, P2, P3 MR 2 Q = 12 l/s P1 MR 3 Q = 14 l/s P1 MR 4 Q = 16 l/s P1 MR 5 Q = 18 l/s P1 MR 6 Q = 20 l/s P1, P2, P3 MR 7 Q = 30 l/s P1, P2, P3 MR 8 Q = 40 l/s P1, P2, P3 MR 9 Q = 50 l/s P1, P2, P3 percentage Effect of clogging of clogging Q h p C0 no clogging 0.00 % 20 l/s P1 C1 clogged bottom slots 0.78% 20 l/s P1 C2 clogged bottom slots and 29.72% 20 l/s P1 1st & 2nd slot row C3 clogged bottom slots and 43.77% 20 l/s P1 1st, 2nd and 3rd slot row C4 clogged bottom slots and 58.66% 20 l/s P1 1st, 2nd, 3rd and 4th slot row Figure 5. Physical model results: dependency curve. surface flow results. The intercepted flow rates Q I depending on the longitudinal slope and surface runoff with constant transverse slope and grate type vary between approximately Q I ≈ 2.95 l/s and Q I ≈ 16.00 l/s. It is expected, that the intercepted flow will converge to a maximum value with increasing surface runoff (not tested yet). As it can be seen in Figure 5, the influence of the longitudinal slope is very low. Having high longitudinal slopes like S L = 10% the bypass flow is small due to higher flow velocities and a smaller water spread width on the street than having small longitudinal slopes. But, the higher the flow velocities, the more water is flowing over the inlet. Both effects cancel nearly each other out, then, the influence of the longitudinal slope is nearly negligible. Due to longitudinal slopes up to 10%, only supercritical flow conditions occur with flow depths up to 3 cm and flow velocities of approximately 1–2 m/s.

Figure 6. Water elevations depending on the inflow without

clogging effects and a low pressure head P1.

Figure 7. Pipe Discharge depending on the water depth in

the street inlet.

3.2 Effect of pressure head h p

Using bidirectional coupled models to simulate the

interaction between major and minor system, the

capacity of the underground system to cope with

the intercepted surface flow has to be known. With no

clogging effects (C0) and a low pressure head in the

pipe system (P1), the capacity of the sludge bucket is Figure 8. Pipe Discharge depending on the water level difference dh. sufficient to drain the intercepted water without backwater effects in the street inlet. The resulting water elevations in the street inlet h i depending on the inflow Q I are displayed in Figure 6 and 7. The pipe flow Q P is the same as the inflow Q I . Hence, with pressure heads below P1 all of the intercepted water can be drained. With an increasing pressure head at the outer boundary (connection pipe) surcharging conditions appear. In Figure 7 the resulting pipe discharges with the according water depth in the street inlet are displayed. With an increasing water level in the manhole up to the defined pressure head P2 the water depths in the street inlet increase. With an inflow more than Q I = 20 l/s the pipe discharge is less than the inflow discharge. The water level starts to increase more rapidly. However, the surface level is barely reached, no surface flooding caused by the underground drainage system occurs. When the water level in the manhole reaches the level of the upper border of the sludge bucket and an inflow of more than 20 l/s is achieved the water enters the surface. Therefore, no results are displayed in Figure 7. The water level difference dh is defined as: with h i = water elevation in the inlet [m] and h p = water elevation in the manhole [m]. In Figure 8 the dependency of dh and the pipe discharge Q P is given. With a pressure head at the level of the bottom of the inlet (P1) the water level h i as well as the water level difference dh increase linearly with increasing discharge – no surcharging conditions appear. 3.3 Effect of clogging With a low pressure head in the pipe system (P1) and an inflow of Q = 20 l/s four different amounts of clogging of the sludge bucket were investigated (Figure 9). In Figure 10 the calculated water elevations resulting in the street inlet are displayed. Even with a nearly fully clogged bucket (60% of the slots are clogged), the remaining openings are efficient enough

Figure 9. Clogging effects.

Figure 10. Water elevations depending on the clogging amount with Q = 20 l/s and P1.

to drain the inflow of 20 l/s without causing surface flooding.

4 CONCLUSIONS

In order to develop new methodologies to manage urban flooding, several simulation runs were done. In previous physical model test runs, the capacity of grate inlets was defined. Depending on the longitudinal and transverse slope of the street, only

Effects of flow orientation on the onset of motion of flooded vehicles

C. Arrighi & F. Castelli

Department of Civil and Environmental Engineering, University of Florence, Italy

H. Oumeraci

Leichtweiss Institute for Hydraulic Engineering and Water Resources, Technical University of Braunschweig, Germany

ABSTRACT: Flood risk in urban areas is recognized as a crucial issue as the population living in cities is constantly increasing with a consequent rise in the value of exposed asset. It has been demonstrated that, in developed countries, most of the fatalities during a flood occurs inside vehicles. In fact, the combination of water depth and velocity acts so that a car can be swept away also for very low water depth. This work aims at identifying the critical conditions for vehicles incipient motion in order to support flood risk mapping and

management and to promote people's education. The dimensionless mobility parameter θ_V derived in a previous study is modified to account for the effect of different flow orientations. The dimensionless mobility parameter θ_V accounts for both flood and vehicle characteristics, thus it overcomes the scatter of diagrams showing the critical dimensional pairs of water depth and velocity. 3D numerical simulations in the OpenFOAM framework have been carried out to understand the role of the angle of incidence of the flow with respect to the car. The hydrodynamic forces are evaluated for different flow regimes, both subcritical and supercritical. The results of the simulations highlight that not only the area exposed to the flow matters for the onset of motion, but also the flow field under the vehicle planform. Existing flume experiments on small-scale models are used for the numerical model validation and as a reference for the interpretation of the numerical results.

1 INTRODUCTION

Nowadays floods cause each year million euros of damages and affect a large number of people world wide. The Flood Directive 60/2007/EC requires the European countries to implement flood hazard and flood risk maps in order to properly managing the risk of inundation, through structural and non-structural mitigation measures. Among the damages a flood can produce, there are fatalities and injuries to people. It has been demonstrated that the majority of fatali ties occurs in vehicles (Jonkman and Kelman 2005; Maples & Tiefenbacher 2009; Fitzgerald et al. 2010), which can be swept away by floodwaters even for low water depths and may turn into deadly trap for the driver and the passengers.

Although people safety is the primary aim of any flood risk management strategy, the studies on vehi cles instability under water flow are sparse and existing hazard criteria suffer from many drawbacks and are not reliable for the application to urban flood scale. In particular, a large scatter affects the existing experi mental critical conditions (Xia et al., 2011; Shu et al., 2011; Xia et al., 2014), which lead to the onset of motion. In fact, the loss of stability in floodwaters not only depends on flood characteristics (i.e. water depth and velocity) but also on the geometric and physical properties of the immersed object. The aim of the work is to introduce a new hazard parameter accounting for both flood and vehicles characteristics and capable of being easily applicable to inundation scale in order to map the likelihood of a vehicle of being swept away during a flood. 2 MOBILITY PARAMETER 2.1 Incipient motion conditions of a flooded vehicle The two recognized hydrodynamic mechanisms by which the stability of a stationary vehicle (i.e. parked) is lost are floating and sliding.Assuming that a car cannot be filled quickly by floodwater, its density is much smaller than water density and most of the weight is usually distributed in the lower front part (close to the engine in the majority of modern cars). The floating instability occurs when the buoyancy and lift effects exceed the weight of the car. The sliding instability occurs when the drag force exceeds the resistance force (i.e. tyre/road friction).These two mechanisms interact as the effect of buoyancy and lift reduce the normal component of the weight thus promoting sliding conditions even for very low water depths. The incipient motion of flooded vehicles can be approached with some similarity to the study of sediment transport in rivers, also taking into account the peculiar features of the vehicles. The main parameters of a vehicle, which influence its incipient motion are: shape, specific weight, weight distribution, position of the chassis over the channel bed, degree of submergence.

Parked cars can be parallel to the longitudinal axis of the street or may have different orientations depending on the geometry of the parking lot. Thus, the effect of flow orientation has to be taken into account in order to better understand the mechanisms underlying incipient motion of vehicles.

The mobility parameter defined by Arrighi et al. (2015) has to be modified to include the orientation of the vehicle with respect to the flow direction. The most evident effect of the angle of incidence is a change in the area of the vehicle, which faces the flow and thus contributes to the drag force.

The forces acting on a partly submerged vehicle are: weight W , buoyancy B, drag D, lift Li and friction. Incipient motion for sliding occurs when the drag force just exceeds the friction force, which is the product of the friction coefficient μ and of the effective weight of the car (Eq. 1)

Since the expression of the hydrodynamic forces requires a reference area, two different surfaces are defined for drag and lift forces. The reference area for drag force is selected equal to the full area of the vehicle projected normally to the flow. The selection of the reference area is arbitrary and the full normal area $(l \cdot \cos\beta + L \cdot \sin\beta) \cdot (H_V - h_c)$ has been preferred to the wetted area $(l \cdot \cos\beta + L \cdot \sin\beta) \cdot (H - h_c)$. Firstly, because the actual water depth on the car surface is different from the undisturbed water depth H, secondly, it significantly varies along the car surface as a consequence of the local vehicle-flow interaction, whose magnitude depends on the flow regime. Thus, the drag force D is defined as

where ρ is the water density, C_D is the drag coefficient, U is the mean flow velocity, l is the car width, L is the car length, β is the angle of flow incidence, H_V is the height of the car and h_c is the elevation of the chassis (see Fig. 1).

The weight of the car is given by

where ρ_c is the car density and g the acceleration of gravity.

Buoyancy and lift force are defined as

where H is the water depth and C_L is the lift coefficient.

The reference area for lift force is the planform area of

Figure 1. Geometric scheme for a vehicle with angle of flow

incidence β. the vehicle.After the substitution of Equations 2–5 into Equation 1 and after some manipulation the following is obtained where C includes the coefficients for drag, lift and friction force and the effect of the angle of flow incidence α (defined in Eq. 10) Fr^2_V is the square of the Froude number of the vehicle and θ_V is defined as the dimensionless mobility parameter for the vehicle where α a correction factor accounting for the flow orientation (i.e. $0^\circ \le \beta \le 90^\circ$) assuming the hypothesis of the existence of two axis of symmetry of the car. 2.2 Dimensionless instability diagram In order to verify the mobility parameter (Eq. 9), the experimental data by Xia et al. (2011, 2013) and Shu et al. (2011) on the threshold of vehicle instability investigated for three orientation angles (0°, 90°, 180°) are considered. The mentioned experimental datasets

Figure 2. Dimensionless instability diagram "mobility parameter vs Froude number" for a flooded vehicle, the areas above and beneath the θ_{Vcr} curve represent stable and unstable conditions respectively.

are about scale 1:43 vehicle models (Xia et al., 2011), scale 1:18 vehicle models (Shu et al., 2011) and scale 1:14 vehicle models (Xia et al., 2014). The experimen tal critical pairs of water depth and velocity (H, U) and the geometry data of the vehicle models allow calcu lating the parameter θ_V, which is represented against Froude number in Figure 2.

The dimensionless parameter θ_V calculated for the available experiments identifies a unique threshold of incipient motion (black line in Fig. 2), for dif ferent vehicle models, scales and flow orientations, thus overcoming the scatter of dimensional instability diagrams.

The threshold critical curve θ_{Vcr} separates stable

conditions (above the curve) from unstable conditions

(beneath the curve)

where a = 8.2, b = 14.1 and c = 5.4, with $R^2 = 0.79$

and RMSE = 16.1.

3 3D NUMERICAL MODEL

3.1 Model set up

For the numerical simulations, the CFD toolbox Open

FOAM is used since it is suitable for numerical

modelling of wide number of applications in coastal

and hydraulics engineering (www.openfoam.org)

including capabilities for wave/current genera

tion/absorption, mesh manipulation and turbulence

modelling. Particularly, the solver waveFoam included

within the library waves2Foam (developed by Jacob

sen et al., 2012 within the OpenFOAM framework)

is adopted. This solver is able to calculate velocity

and water depth with Reynolds Averaged Navier

Stokes equations for two incompressible, isothermal,

immiscible fluids capturing the fluid-fluid interface

through the volume of fluid (VOF) method. The solver applies the relaxation zone technique for current generation together with absorption of its reflection (due to obstacles within the computational domain). This 'active sponge' layer allows avoiding interferences with the inflow boundary condition. The relaxation zone technique implemented in the solver is based on an extension to that of Mayer et al. (1998). Among the car types used in the flume experiments, we choose the Ford Focus for the numerical simulations because it is representative of a class of medium city cars. To generate the mesh around the car model the triangulated geometry of the vehicle, provided by LaSIS laboratory was used. A mesh sensitivity and turbulence model sensitivity analysis have been carried out to select a compromise between accuracy and computational time (Arrighi et al., 2015). The snappyHexMesh tool allows refining the mesh close to the car surface (cell size is set to 0.03 m) while in the whole mesh domain the maximum size is 0.25 m. The total number of cells is around $5 \cdot 10^5$. In the numerical study by Arrighi et al. (2015) two flow orientations (0° and 180°) reproducing experimental incipient motion conditions (Shu et al., 2011) were investigated. In this work the same pairs of water depth and velocity have been set as boundary conditions for two new flow orientations ($\beta = 65^\circ$ and 90°, see inset in Fig. 3) in order to understand the effect of the angle of flow incidence on the hydrodynamic forces. Drag and lift forces calculated through pressure integration on the car surface include both pressure and viscous forces acting in the flow and vertical direction respectively. However, the contribution of the viscous forces is negligible with respect to pressure forces (they differ of six-seven orders of magnitude). 4 NUMERICAL RESULTS The representation of the dimensional forces on the vehicle underlines the large difference in terms of impact for the same pairs of water depth and velocity and different flow orientations. Both sub-critical and supercritical flow regimes have been investigated, with water depth ranging from 0.16 m up to 0.5 m and flow velocity ranging from 6.0 m/s and 0.8 m/s (scaled to prototype values through Froude similarity). Figure 3 (top) shows that the drag force for $\beta = 0^\circ$, $\beta = 180^\circ$ is smaller than the drag force for $\beta = 90^\circ$, $\beta = 65^\circ$. While for sub-critical flow conditions the values of drag force are comparable for all the flow orientations, for super-critical flows the drag force for $\beta = 90^\circ$, $\beta = 65^\circ$ reaches twice the value for the same simulated conditions (H,U) and flow orientations $\beta = 0^\circ$, $\beta = 180$. This means that a lateral flow impact might be more critical for the onset of vehicle motion. The drag force is about 6000 N for the simulated $\beta = 65^\circ$, H = 0.23 m and U = 5.09 m/s (i.e. Fr = 3.4). Figure 3 (bottom) compares the lift force versus Froude number, calculated for the different flow orientations. Lift forces for$\beta = 90^\circ$ and$\beta = 65^\circ$ (square and diamond symbols) appear strongly different from the

Figure 3. Comparison between drag (top) and lift (bottom)

forces versus Froude number for different flow orientations.

$\beta = 0^{\circ}$ and$\beta = 180^{\circ}$ simulated cases. In fact, lift forces for $\beta = 90^{\circ}$ and $\beta = 65^{\circ}$ are lower than lift forces eval uated for $\beta = 0^{\circ}$ and $\beta = 180^{\circ}$ for the same simulated pairs of water depth and velocity.

In particular, lift forces are negative for Froude number in the range 1.6-4. This means that the ver tical force is directed downward for the mentioned flow conditions, due to a suction effect beneath the car planform and thus it contributes to the global sta bility of the vehicle. The suction effect has been also experimentally observed by Bonham and Hattersley (1967).

The hydrodynamic forces evaluated by the 3D numerical model have been validated through the com parison with the experimental data on a Ford Focus scale 1:18 with $\beta = 0^{\circ}$ (Shu et al., 2011) in Arrighi et al., (2015). Thus, the model has been proven suit able for calculating the hydrodynamic forces. The analysis of the forces balance for sliding as in Eq. 1 shows that a vehicle oriented laterally with respect to the mean flow velocity (e.g. 65° and 90°) can be more easily mobilized by floodwaters with respect to $\beta = 0^{\circ}$ and $\beta = 180^{\circ}$ orientations, although the differ ence between the flow orientations is not as significant as the difference observed in the estimated drag force

(Fig. 3 top). In fact, the negative lift force for β = 65 ◦

OpenFoamr, International Journal for numerical methods

in fluids, 70, 1073–1088.

Jonkman, S.N., Kelman, I., 2005. An analysis of the causes

and circumstances of flood disaster deaths. Disasters, 29,

75–97.

Maples, L., Tiefenbacher, J., 2009. Landscape, development,

technology and drivers: The geography of drownings

associated with automobiles in Texas floods, 1950–2004.

Applied Geography, 29, 224–234.

Mayer, S., Garapon, A., Sørensen, L.S., 1998. A fractional

step method for unsteady free surface flow with applica

tions to non-linear wave dynamics. International Journal

for Numerical Methods in Fluids, 28, 293–315. Shu, C., Xia, J., Falconer, R. & Lin, B., 2011. Estimation of incipient velocity for partially submerged vehicles in floodwaters. Journal of Hydraulic Research, 49: 709–717. Xia, J., Teo, F. Y., Lin, B., Falconer, R., 2011. Formula of incipient velocity for flooded vehicles. Natural Hazards, 58, 1–14. Xia, J., Falconer, R.A., Xiao, X. & Wang, Y. 2014. Criterion of vehicle stability in floodwaters based on theoretical and experimental studies. Natural Hazards, 70: 1619–1630. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Development of a computationally efficient urban flood modelling approach

V. Wolfs, V. Ntegeka, D. Murla Tuyls & P. Willems

Hydraulics Division, Department of Civil Engineering, KU Leuven, Leuven, Belgium

ABSTRACT: This paper presents a parsimonious and data-driven modelling approach to simulate urban floods.

Flood levels simulated by detailed 1D-2D hydrodynamic models can be emulated using the presented conceptual

modelling approach with a very short calculation time. In addition, the model detail can be adjusted, allowing the

modeller to focus on flood-prone locations. This results in efficiently parameterized models that can be tailored

to applications. The simulated flood levels are transformed into flood extent maps using a high resolution

(0.5-meter) digital terrain model in GIS. To illustrate the developed methodology, a case study for the city of

Ghent in Belgium is elaborated. The configured conceptual model mimics the flood levels of a detailed 1D-2D

hydrodynamic InfoWorks ICM model accurately, while the calculation time is an order of magnitude of 10 6 times

shorter than the original highly detailed model. The proposed models can be used for numerous applications

of urban water management requiring fast models, such as early warning systems, uncertainty analyses and

optimization, e.g. to determine real-time storage operations.

1 INTRODUCTION

Urban flooding causes worldwide significant disrup

tion to society, huge economic losses and imposes

serious health risks. The rapidly increasing urbaniza

tion, aging storm networks and climate change will

increase urban flood hazard and risk. Recent studies

on future climate change impacts show that extreme

rainfall intensities may strongly increase (Willems

et al., 2012; IPCC, 2014) and as a result also the urban drainage flows and flood hazard (Willems, 2013). Increasing the resilience of urban areas to local rainfall-induced floods is therefore a major objective of present and future water management.

Next to the design of adaptation measures to reduce the flood hazard, measures that increase the prepared ness of disaster agencies and the self-coping capacity of people becomes more and more important. One of the latter type of actions is the setup of an urban flood forecasting and warning system. For forecasting of floods in urban areas as a result of extreme rain storms, called pluvial floods, the forecasting has to focus on extreme convective rain storms. Given their short dura tion but also because of the quick responses of urban drainage systems to rainfall, forecasting of convective rain storms and related pluvial floods can only be done for short lead times. This type of forecasting is called "nowcasting".

One such operational nowcasting system is the operational Short-Term Ensemble Prediction System (STEPS), which was originally co-developed by the UK Met Office and Australian Bureau of Meteorology (Bowler et al., 2006). The system has recently been adapted for Belgium (STEPS-BE). STEPS-BE is based on a 4 C-band radar composite as input and provides rainfall forecasts at high resolution (1 km/5min) with 20 ensemble members and a lead time of up to 2 hours (Foresti et al., 2016). As the forecasts are updated every few minutes, it is essential that pluvial inundation models are developed to suit operational forecasting requirements. More specifically, fast models are required such that the real-time flood impact simulations can be conducted every few minutes. Because uncertainties may be significant, especially for longer lead times, such uncertainties also have to be accounted for in the forecasting system. Since such quantification requires multiple runs, e.g. simulating the impacts of ensemble rainfall predictions, incorporating models with very short calculation times becomes even more important. In addition, the models need to be accurate, and “integratable” in existing systems. Given the specific characteristics of urban floods, high resolution models moreover are required to represent topography, the presence of houses, inlets, etc. Various model types are combined and used to simulate urban floods (see e.g. Henonin et al. (2013) and René et al. (2014) for a comprehensive discussion). Although physically-based white box models are the most popular model type in urban drainage modelling (Parkinson and Mark, 2005), coupled 1D-2D models are commonly applied for urban flood studies (e.g. Schmitt et al., 2004; Carr & Smith, 2006). In such a setup, the underground drainage network is modelled in 1D, while the surface flow model is 2D. Such detailed 2D approach is necessary to accurately represent the full city surface. However, a 1D-2D approach is computationally very expensive and therefore does not meet the requisite functional model characteristics of flood forecasting systems and many other urban flood applications requiring fast models.

This paper tries to overcome this issue by introducing a new and parsimonious modelling approach for simulating urban flood water levels that is compatible with the “CMD” framework for efficient hydraulic modelling of sewers (Wolfs & Willems, submitted). A new module is introduced to emulate urban flood lev els generated by highly detailed 1D-2D hydrodynamic models. The predicted water levels are translated to flood maps by means of GIS procedures using data from a high resolution digital terrain model (DTM). The approach is highly flexible and allows the user to focus on flood-prone areas. The proposed method and framework can deal with different temporal and spatial scales, and can for instance use (weather) radar data as input. A similar approach was already developed for river systems (Wolfs et al., 2015) and applied for var ious applications (e.g. De Vleeschauwer et al., 2014; Vermuyten et al., 2014; Wolfs & Willems, 2015).

To illustrate the developed approach, it was imple mented for the case study of the sewer system of the city of Ghent, Belgium. It starts from a highly detailed 1D-2D InfoWorks ICM model to simulate urban floods at high spatial resolution. A fast conceptual hydraulic model was identified and calibrated to that detailed model. The conceptual model will then be applied in a surrogate, complementary way. It aims to accurately emulate water level simulation results of the detailed ICM model. Water level simulation results of the con ceptual model are used as input for the GIS-DTM based flood mapping. The derived conceptual model is several orders of magnitude faster than the original

1D-2D hydrodynamic model, and is ideally suited to be used in real-time flood forecasting systems. First, the methodology is presented, followed by a discussion of the case study area and available data. Next, the results of the case study are presented and discussed. Finally, conclusions and a view on future developments are given.

2 METHODOLOGY

The main objective of this research is to develop and test fast models that can accurately calculate instan taneous urban flood levels and visualize the flood extent. Simulating urban flood levels is very compli cated due to the large number of interacting processes. Flood levels mainly depend on the hydraulic state of the subsurface urban drainage network, the topogra phy, and interactions between the underground system and surface. Hence, in order to accurately calculate urban flood levels, the surface flow should also be simulated, which in turns necessitates spatially highly

detailed maps of the topography. However, simulating Figure 1. Schematization of the modelling approach. such detailed models is computationally very expensive.Therefore, we suggest using mechanistic, fast and parsimonious conceptual models that aim to emulate urban flood level data sets. By emulating such data sets, the surface runoff does not have to be simulated explicitly, since the effect of the surface flow is inherently present in the data set. This strongly limits the calculation time, making these models suitable for flood forecasting systems and other applications requiring fast models. A three step approach is proposed to set up such models. Figure 1 illustrates the proposed methodology. In the first step, accurate flood level measurements are gathered to configure the model. Due to the lack of accurate level measurements, simulation results of hydrodynamic models are used in this study to develop and test the approach. The use of these “virtual” measurement sets is often advocated in literature to compensate for such data shortage in urban drainage modelling (e.g. Vaes, 1999; Meirlaen et al., 2001). Since urban flood levels are determined by fine-scale processes and due to the close interaction with the underground drainage network, highly detailed 1D-2D white box model must be employed to ensure realistic urban flood levels are obtained. In the second phase, a parsimonious and fast model is configured to simulate the urban flood levels at selected locations. This model is set up using the simulation results obtained in the previous step. Note that the use of simulation results instead of in situ measurements to calibrate and validate the model does not put stringent limitations on the use and accuracy of the developed approach. Due to the data-driven character of conceptual emulation models, accurate and a sufficient amount of data that cover a wide range of system

dynamics are essential to ensure proper configuration

of these models. The presented approach is not tai

lored to emulate results generated by a specific model

type or software package, but aims to be generically

applicable.

Finally, the urban flood level predictions are trans

lated to flood extent maps using GIS procedures.

The following sections elaborate on the second and

third steps of the methodology.

2.1 Fast model to simulate urban flood levels

The proposed approach tries to emulate urban flood

levels using a fast parsimonious model that does not

model the surface flow during floods explicitly. By

obviating such costly calculations, the calculation time can be significantly reduced compared to full hydro dynamic 1D-2D models. In order to configure such parsimonious models successfully, three critical crite ria should be met. Firstly, the flood level set used to calibrate and validate the model should be accurate and realistic. The set should thus inherently account for the complex processes that influence the water level. Secondly, there must be a functional relation ship between the sought flood levels and several states in the system that can be calculated accurately. Thirdly and finally, this relationship should be identified and parameterized using one or more model structures.

By using simulation results of a hydrodynamic model, it is assumed that the first condition is satis fied. The detailed 1D-2D hydrodynamic model should be able to generate sufficiently accurate flood lev els. A careful selection of several system variables is necessary to comply with the second condition.

The hydraulic state of the sewer system, such as the degree of filling prior to a storm event, has a major impact on floods. It is crucial to account for such (antecedent) conditions. It seems plausible to assume a dependency between the volume of part of the sewer system and the flood level at a selected location. To simulate such volumes quickly and accurately, the con ceptual hydraulic modelling framework developed by Wolfs & Willems (submitted) is used. This framework is denoted as “CMD” and can account for various ele ments that influence the flow and thus volumes in the sewer system, including backwater effects, reverse and pressurized flows and controllable structures such as pumps. Thus, the volume of part of the sewer sys tem is used as first input. It is important to note that CMD allows varying levels of model detail. Thus, the modeller could use the volume of a single conduit up to the volume of an entire sewer system as input. The most appropriate level of lumping is determined using a trial-and-error procedure. Rainfall remains the main driver of urban floods. Therefore, the rain fall intensity (averaged over the response time of the system) is used as second input variable for simulat ing the flood level. Alternatively, the rainfall runoff flow could also be used as input, since such flow already accounts for antecedent surface conditions.

Rainfall runoff flows are calculated in CMD using the Wallingford Model, which is by default used in the hydrodynamic InfoWorks software (Innovyze, 2014). Hence, it is assumed that there exists a functional relation between the two selected inputs, namely the averaged rainfall intensities and volume in selected areas of the sewer system, and the urban flood level at a specified location. Finally, a model structure must be identified and calibrated to ensure that flood levels can be simulated using these two selected inputs. Given the large number of interacting processes, the use of machine learning techniques is proposed. These techniques are very flexible and can configure themselves using supervised learning to emulate complex data sets. The modelling approach incorporates a serial connection of two artificial neural networks (ANNs). The first is a binary classification ANN to determine if there is urban flooding at the selected location. This ANN has a single hidden layer with an adjustable number of nodes and uses the two selected inputs. It relies on sigmoid and softmax transfer functions in the hidden and output layers respectively, which are trained via a scaled conjugate gradient optimization to minimize the cross-entropy. This set-up improves classification performance. The outcome of the ANN is the probability at every time step during a simulation of having urban floods (i.e. the water level surpasses the ground level). If this probability exceeds a specified threshold, the second ANN is triggered to calculate the flood level. This second ANN uses a sigmoid and purely linear transfer functions in the hidden and output layers and are trained via Levenberg-Marquardt optimization using Bayesian Regularization and early stopping. To improve generalization and to limit the influence of the initially randomized parameters of the ANN, an ensemble ANN is used by averaging the outcome of several trained ANNs. After configuring both ANNs, the threshold probability that activates the second ANN is optimized by minimizing the rootmean-square error (RMSE). To facilitate and speed up ANN configuration, the set-up procedure was programmed in MATLAB using the Neural Network Toolbox. This data-driven yet mechanistic approach inherently accounts for the complex dynamics and interactions between the sewer system and surface flow that are present in the calibration set. Given the data-driven character of the approach, having a sufficient amount of data that includes several urban flood events is a requisite for configuring accurate models with good generalization capabilities. 2.2 Flood mapping Next to the sewer hydraulic computations, also the 2D surface inundation modelling and mapping are constrained with computational resources. Real-time urban flood nowcasting demands flood mapping techniques that are fast enough. In this regard, researchers and practitioners have developed GIS flood inundation techniques with minimal data requirements

(Zhang & Pan, 2014; Sampson et al., 2012). For

this study, flood mapping is based on a simplified

methodology that generates flood extents in a rela tively fast way based on flood levels and topographic information. The methodology is akin to the volume spreading algorithm applied in ISIS-FAST (Néelz & Pender, 2010), albeit with a focus on depth spreading. The steps involved in the methodology are described henceforth.

The first step involves the generation of a detailed topographic map of high resolution (1m or less). Steps should be taken to ensure that errors within the map are corrected to minimize the effects of data errors in the final flood extent maps.

In the second step, catchment zones with depres sions (pits) are defined. Pits are those locations where all surrounding cells have flow directions point ing towards them. The identification of pits is done through an iterative approach with the use of param eters for storage volume, area and depth. However, not all pits are relevant as some pits are considered artificial. Therefore validation against observed flood extents or other sources of flood inundation is required. Without data on observed flood extents for urban inundation, results of 1D-2D inundation models help to locate regions that are flood prone and pinpoint locations of depressions. From these locations, better parameters for more realistic pits are estimated.

The third step involves the selection of control points for flood level extension. These locations do not necessarily have to be at pits but should fall within the catchment boundaries. For 1D-2D simulations, water level profiles along sewer pathways in the vicinity of flooded locations provide a suitable basis for selec tion. For instance, locations where the hydraulic grade line is close to the 2D water depth would be more appropriate for flood extension. For flood extents with more than one catchment or depression, a selection of different locations is done.

The fourth step involves the extension of the water levels based on the DTM. The assumption is that if the ground elevation within a depression is lower than the flood level, then the grid cell is considered flooded if it is topographically connected to the refer ence flood cell, which is the control point. Repeating this over all grid cells within a depression gives a flood extent map.

3 CASE STUDY AND AVAILABLE DATA

The developed approach was applied and tested for the urban drainage system of the village of Oostakker, a district of the city of Ghent in Belgium (Figure 2). A detailed 1D-2D full hydrodynamic model of the sewer network (1D) and the surface (2D) was setup, imple mented in InfoWorks ICM. This ICM model covers

the entire sewer system of the districts of Oostakker

and Sint-Amandsberg. It counts in total 6025 con

duits, 182 hydraulic structures (such as pumps, weirs,

sluices and orifices) and 5855 manholes. The system Figure 2. Detailed 1D-2D InfoWorks ICM model of the case study area (left) and hydraulic conceptual CMD model (right). releases water to receiving water bodies via 39 outfall nodes. This detailed 1D representation of the sewer network is coupled with a surface inundation model at high resolution based on a unique very high resolution DTM available for the study area. The surface inundation model makes use of nested triangular meshes at different resolutions; these include streets (3.75– 15 m 2), high flood hazard areas (12.5–50 m 2) and low flood hazard areas (75–300 m 2). The mesh zones are generated from a 0.5x0.5x0.05m DTM (AGIV, 2015) with buildings defined as no flow zones. Additionally, land use areas are classified according to the dominant surface cover type into water, pervious and impervious areas. The interaction between the 1D underground sewer conduits and the 2D surface was through 2D manholes. A 2D manhole is conceptualized as a weir with a crest level taken as the ground level and a crest length equal to the shaft circumference of the node (Innovyze, 2014). In this way, water is exchanged to the 2D surface when the pressure head at the manhole exceeds the ground level. Figure 3 illustrates this approach for a subzone of the study area. The street with the highlighted nodes ‘SR07994104’and ‘SR07995401’is prone to flooding. The nodes shown in the map are the water level calculation nodes in the InfoWorks ICM model. The node ‘SR07994104’represents the highest situated manhole of the sewer system and floods first. From this location, water flows over the surface to other nodes in the street. The node ‘SR07995401’ represents a manhole which is situated in a depression in the topography. Hence, the flow at this location is both induced by sewer floods through the manhole at that specific location, but also by surface flows from other nodes. By selecting two nodes, it is possible to assess the performance for both types of flooding. Six spatially uniform synthetic storm events with different frequencies of occurrence of 10 (denoted as ‘f10’) and 7 (‘f07’) times per year, and return periods of 2, 5, 10 and 20 (respectively ‘t02’, ‘t05’, ‘t10’ and ‘t20’) years were simulated. These six events lead to different flood levels. All models are calibrated for the ‘f10’, ‘t02’, ‘t10’ and ‘t20’ events, and validated for the ‘f07’ and ‘t05’ events.

Figure 3. The most flood prone subzone of the study area, indicating the pipes of the 1D sewer network and the calculation nodes in the InfoWorks-ICM model, and the 2D triangular mesh zones. Buildings (grey) are also shown.

4 RESULTS

4.1 Hydraulic model

A conceptual hydraulic model of the sewer system was configured based on simulation results of the detailed hydrodynamic models of the six synthetic storm events. This conceptual model is configured to simulate the volumes in the sewer system, which are in turn used as input to simulate the flood levels (see §2.1 and §4.2). The conceptual hydraulic model divides the entire sewer system in six interconnected reservoirs (see Figure 2). The average Nash-Sutcliffe efficiencies (NSE; Nash and Sutcliffe, 1970) for the simulated volumes in these six cells for the calibration and validation events are 0.94 and 0.93 respectively. An NSE of unity indicates a perfect match between the conceptual and hydrodynamic models. Hence, the obtained NSE values indicate that the conceptual model manages to emulate the hydraulics and volumes of the detailed hydrodynamic model accurately. A comprehensive discussion of the calibration and validation results can be found in Wolfs and Willems (submitted).

4.2 Flood levels

Next, the two ANNs in series are configured for both investigated locations to calculate when flooding occurs and, if relevant, the flood levels. First, the binary neural network is trained according to the procedure outlined in §2.1. Water levels exceeding the flood level are given target values of unity, while others are zero. After training, the obtainedANN is visualized in a grid (see Figure 4a for the binary classification ANN and the target values for location 'SR07994104'). Target values where flooding occurred in the hydrodynamic ICM model are marked in red, while couples without flooding are shown in green. It is obvious that there is a clear segregation of the 2D input space possible into a subspace with no flooding and a subspace where the water level overtops the ground level, leading to floods. Next, the second ANN is configured, aimed to predict the magnitude of the flood levels. Only water levels above ground level are used as training data to ensure that the ANNs can purely focus on emulating the magnitude of the flood levels. An ensemble of four networks is trained. The obtained ensemble of ANNs is then translated to a single entity using simple averaging. By using such an averaged ensemble, the generalization capability is improved, since the variability of each ANNs' response is reduced. Finally, possible negative values in the response domain (thus representing water levels below the surface) are converted to zero. Note that negative values in the ANN's outcome will rarely occur, since the provided training values are strictly possible. Indeed, this second ANN is only being calculated when the first (binary) ANN indicates that flooding occurs. The obtained network for location 'SR07994104' is shown in Figure 4b. The ANN surface response misses only very few targets. In the third configuration step, the threshold probability that is used to determine precisely when flooding occurs is optimized. Note that the training of these ANNs only takes a few seconds due to the use of solely two inputs (volume and averaged rainfall intensity) and the low number of nodes in the hidden layer (≤10). Figure 5 shows the simulated flood levels for the four events that lead to flooding. It is clear that the conceptual model manages to emulate the results of the hydrodynamic ICM model very accurately. Table 1 summarizes the RMSE and NSE which are calculated by comparing the simulation results of the conceptual and 1D-2D ICM models. Note that only the 15-minute interval in which flooding occurs is used to calculate both goodness of fit statistics to ensure that only on the period of interest is focused. The results show that the conceptual model can accurately predict the flood level at both locations, although the accuracy of the 't05' event which is used for validation is lower. However, the deviations remain limited as indicated by the low RMSE value. 4.3 Flood extent Flood extent maps were generated for the flood depth computed by the conceptual model for location 'SR07994104'. However, as the depth for the 2-year event was too shallow (less than 0.05 m) it was not extended because the DTM vertical accuracy is around 0.05 m. This implies that spreading flood depths close to 0.05m is not feasible. From Figure 6, it is evident that flood extents for the different return periods are almost indistinguishable which is explained by the

Figure 4. Trained ANNs for location "SR07994104"; (a)

binary classification ANN which calculates the probability

of flooding; (b) ANN which simulates the flood level in case

the binary classification ANN assesses that flooding occurs.

Figure 5. Flood level simulation results of the conceptual

and 1D-2D ICM model for location 'SR07994104' for the

events leading to flooding.

close flood peak depths shown in figure 4. It is also

apparent that GIS-based flood extents tend to be higher

than the flood extents computed by InfoWorks-ICM,

but the difference is small. Considering the 20-year

flood extent around node 'SR07994104', the GIS

based flood extent is only about 8% higher than the Table 1. Goodness of fit statistics of the water level simulations at the two selected locations for the different events (C = calibration; V = validation). 'SR07994104' 'SR07995401' RMSE [mm] NSE RMSE [mm] NSE f10 (C) 0.0 1.00

0.0 1.00 f07 (V) 0.0 1.00 0.0 1.00 t02 (C) 2.1 0.86 6.6 0.92 t05 (V) 3.5 0.96 33.1 0.34 t10 (C) 2.9 0.99 6.8 0.99 t20 (C) 2.0 1.00 13.0 0.97 Figure 6. Maximum flood extent simulated by the InfoWorksICM for the 20-year storm (Left) and by the GIS-based approach for 5-,10-,20-year storms (Right). flood extent by the InfoWorks-ICM. Highest differences are located in the low–lying areas downstream of the control node. 5 DISCUSSION AND CONCLUSIONS A new parsimonious and fast emulation approach was presented to simulate flood levels in urban areas. Application on a case study showed that the approach can successfully mimic the simulation results of a detailed 1D-2D hydrodynamic InfoWorks ICM model. The approach combines three modules: (1) a conceptual model (set up using the “CMD” framework; see Wolfs and Willems (submitted)) to simulate flows and volumes given rainfall intensities, (2) a serial connection of two ANNs to simulate flood levels at specified locations, and (3) a GIS module to visualize the flood extents. This paper focuses on the latter two components. A connection of two ANNs is configured for each location where flood levels are simulated. The ANNs depend solely on rainfall intensities and the simulated volume of part of the sewer system to calculate the flood level. This volume is simulated by the hydraulic

conceptual model (CMD). Given the flexibility of the CMD conceptual model, lumping of areas can range from vary small scales (i.e. the volume is being simu lated for a combination of several pipes) up to entire districts (i.e. one volume is simulated for an entire district). Naturally, the applicability of the presented approach to simulate floods depends on the presence of a functional relationship between the simulated volume and rainfall intensities, and the sought flood levels. Given the flexibility of the approach, it is impos sible to define a fixed set of crisp criteria to evaluate if such functional relationship exists. Instead, the exis tence of such relationship should be evaluated in an ad hoc fashion. Note that it is easily possible to use other inputs sets as well (e.g. add other variables to the input set, such as flows).

The results of the case study show that the derived set-up (conceptual CMD model extended with ANNs for flood level calculation) can simulate urban floods up to 10 6 times faster than the original 1D-2D hydro dynamic model. This vast speed gain is achieved due to the data-driven character of the approach, which obviates detailed calculations of both the flow in the sewer system and the surface flow. To ensure that the data-driven approach can simulate the dynamics of the sewer and surface flow, the data used to set up the models needs to cover both dynamics. In addition, it is crucial to employ a large data set to configure a model with adequate generalization capabilities.

In a second phase, the urban flood levels were trans formed to flood extent maps using a simple spreading algorithm in GIS based on the very fine resolution DTM. Flood extent mapping based on such algo rithm was found to be a practical alternative to the more detailed 1D-2D full hydrodynamic modelling.

Even though detailed flood models are more pre cise, calculation times remain prohibitive for pluvial forecasting over short time intervals. Nonetheless,

comparing the flood extents simulated by the 1D-2D full hydrodynamic model with the GIS-based extents provides insights on selecting important parameters for defining depressions. The GIS-based technique allows for temporal flood maps, which is benefi cial for understanding flood propagation. However, flood level spreading does not take into account flood volumes, which often lead to an overestimation of inundated areas especially in flat areas, unless this can be accounted for in the conceptual sewer model. Moreover, water movements and time delays at the surface runoff are not account for in the approach. The quality of the DTM obviously is of critical impor tance for accurately describing flood extents based on flood depths. In some cases, some infrastructure may appear flooded but in reality they could be above the flood level. It is crucial to stress that GIS-based flood extent maps generated from depths alone are aimed at quick assessment and should be treated as such.
The models configured with the presented method ology can be tailored to the intended application. Due to their flexibility, the model detail can eas

De Vleeschauwer, K., Weustenraad, J., Nolf, C., Wolfs, V., De Meulder, B., Shannon, K. & Willems, P., 2014. Green-blue water in the city: quantification of impact of source control versus end-of-pipe solutions on sewer and river floods. Water Science and Technology 70(11): 1825–1837.

Foresti, L., Reyniers, M., Seed, A., & Delobbe, L., 2016. Development and verification of a real-time sto-chastic precipitation nowcasting system for urban hydrology in Belgium. Hydrology and Earth System Sciences 20: 505–527.

Henonin, J., Russo, B., Mark, O. & Gourbesville, P., 2013. Real-time urban flood forecasting and modelling – a state of the art. Journal of Hydroinformatics 15(3): 717–736.

Innovyze, 2014. InfoWorks ICM Help v5.5.

IPCC 2014. Climate Change 2014: Impacts, Adaptation, and Vulnerability. Summary for Policymakers, Working Group II contribution to the Fifth Assessment Report of the Intergovernmental Panel on Climate Change, 44 p.

Meirlaen, J., Huyghebaert, B., Sforzi, F., Benedetti, L. & Vanrolleghem, P., 2001. Fast, simultaneous simulation of the integrated urban wasewater system using mechanistic surrogate models. Water Science and Technology 47(7): 301–308.

Nash, J.E. & Sutcliffe, J.V., 1970. River flow forecasting through conceptual models part I – A discussion of principles. Journal of Hydrology 10(3): 282–290.

Néelz, S. & Pender, G., 2010. Benchmarking of 2D hydraulic modelling packages. UK Environment Agency, Bristol,

UK.

Parkinson, J. & Mark, O., 2005. Urban Stormwater Man

agement in Developing Countries. 2nd edition, IWA

Publishing, London.

René, J.-R., Djordjevic, S., Butler, D., Madsen, H. & Mark,

O., 2014. Assessing the potential for real-time urban

flood forecasting based on a worldwide survey on data

availability. Urban Water Journal 11(7): 573–583.

Sampson, C.C., Fewtrell, T.J., Duncan, A., Shaad, K., Horritt,

M.S. & Bates, P.D., 2012. Use of terrestrial laser scan

ning data to drive decimetric resolution urban inundation

models. Advances in Water Resources 41: 1–17. Schmitt, T.G., Thomas, M. & Ettrich, N., 2004. Analysis and modelling of flooding in urban drainage systems. Journal of Hydrology 299: 300–311. Vaes, G., 1999. The influence of rainfall and model simplification on combined sewer system design. PhD Thesis, Department of Civil Engineering, University of Leuven, Belgium. Vermuyten, E., Van den Zegel, B., Wolfs, V., Meert, P. & Willems, P., 2014. Real-time flood control by means of an improved MPC-GA algorithm and a fast conceptual river model for the Demer basin in Belgium. Proceedings of the 6th International Conference on Flood Management. ICFM. Sao Paulo, Brazil, September 2014, pp. 1–8. Willems, P. & Vrac, M., 2011. Statistical precipitation downscaling for small-scale hydrological impact inves-tigations of climate change. Journal of Hydrology 402: 193–205. Willems, P., 2013. Revision of urban drainage design rules after assessment of climate change impacts on precipitation extremes at Uccle, Belgium. Journal of Hydrology 496: 166–177. Willems, P., Olsson, J., Arnbjerg-Nielsen, K., Beecham, S., Pathirana,A., Bülow Gregersen, I., Madsen, H. & Nguyen, V.-T.-V. 2012. Impacts of Climate Change on Rainfall Extremes and Urban Drainage. IWA Publishing, 252 p., Paperback Print ISBN 9781780401256; Ebook ISBN

9781780401263. Wolfs, V. & Willems, P., 2015. Quantification of impact of retention basins on river floods in the Dender catchment in Belgium using computationally efficiënt models. 36th IAHRWorld Congress, Delft/The Hague,The Netherlands, 29 June–3 July 2015. Wolfs, V., Meert, P. & Willems, P., 2015. Modular conceptual modelling approach and software for river hydraulic simulations. Environmental Modelling and Software 71: 60–77. Wolfs, V. & Willems, P., submitted. Conceptual modelling approach and software for sewer hydraulic simulations. Water Resources Management. Zhang, S. & Pan, B., 2014. An urban storm-inundation simulation method based on GIS. Journal of Hydrology 517: 260–268. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Rapid flood inundation modelling in a coastal urban area using a surrogate model of the 2D shallow water equations

L. Cea, M. Bermudez, J. Puertas, I. Fraga & S. Coquerez

Environmental and Water Engineering Group, Departamento de Métodos Matemáticos y de Representación,

Universidade da Coruña, Spain

ABSTRACT: Two dimensional shallow water models have demonstrated good capabilities for flood inundation mapping in urban areas. However, even if High Performance Computing techniques have greatly decreased the computational time needed to run a 2D inundation model, this approach remains unsuitable for applications as real time forecasting or uncertainty propagation in a Monte Carlo context, which requires the evaluation of hundreds or thousands of model runs. For these applications there remains a need for fast urban inundation models. In this paper we propose and compare the application of different linear, non-linear and non-parametric regression techniques as surrogate models of the 2D shallow water equations (SWE) applied to flood inundation

mapping in ungauged urban areas. A coastal urban area is used as a test case. The case is specially challenging since the spatial distribution of water depth in the study area depends on the flow discharge in three different tributaries as well as on the tidal level and range.

1 INTRODUCTION

Physically-based flood inundation models are widely used tools for simulating river hydraulics and flood plain inundation processes. In these models, the flood propagation is generally described by the shallow water equations, which must be solved using an appro priate numerical technique. Models solve either the full or simplified forms of these equations on a grid of a certain resolution and with a chosen time discretiza tion. In general, an inevitable compromise between the accuracy and the efficiency of these models can be found (Chen et al., 2012). Higher accuracy can be obtained by increasing the physical complexity and the spatial and temporal resolution of a model, but at the cost of higher setup and computing time.

In the case of urban areas, accurate numerical mod els require at least a two-dimensional treatment of the surface flow hydraulics and a high enough grid res olution to capture the complex flow paths and local hydraulic effects (Fewtrell et al., 2008; Yu & Lane, 2006). Different model approaches for urban catch ments, based on these premises, have been reviewed by Hunter et al. (2008). All of them provide detailed information about the flooding dynamics, but at the same time require large distributed datasets for their parameterization and validation. Although new data capture techniques have made them more readily available, the processing of these data and the con struction of the model remains time consuming. On the other hand, the computational time step decreases with grid size to ensure model stability. Consequently, the fine discretization needed in urban areas is hardly compatible with fast computation of the flow field. This conventional physically-based approach is hence generally unsuitable for large scale modelling or real-time forecasting in urban areas. It is likewise impractical for applications which involve a high number of model runs.Typical examples of the latter are the analysis of different flood scenarios, alternative mitigation strategies or uncertainty propagation in a Monte Carlo context. Therefore, there remains a need for fast and accurate urban flood models. One straightforward way to meet this challenge is the use of coarser grids and the development of subgrid models to account for topographic variability that is too small to resolve with the computational mesh (Sanders et al., 2008). Within each computational cell of a subgrid model, effects of small-scale solid objects are represented by porosity parameters. This allows the simulation of urban flood flows without the need of a very fine and computationally demanding mesh (Cea & Vazquez-Cendon, 2010; Guinot, 2012; Schubert & Sanders, 2012). In many practical situations, the main concern is making accurate and timely predictions at specific locations, with limited interest in the mechanisms of the flow processes (Chau et al., 2005). In these cases, data driven models can constitute an alternative approach. They are based on relationships manifested in historical records between input and output variables without analyzing the internal structure of the physical process. In recent years, many nonlinear predictors based on artificial intelligence have been applied in the field of hydrological modelling: artificial neural networks, adaptive neuro-fuzzy infer ence systems or support vector machine, to name some of them. These methods have been applied suc cessfully by Anctil & Rat (2005), Bhagwat & Maity (2013), Chang et al. (2010), Chen et al. (2013), Chi ang & Chang (2009), Dawson et al. (2006) or Li et al. (2014) for rainfall-runoff modelling and flow fore casting. Conventional regression analysis, either linear or non-linear, has also been used for similar purposes in the works by Engeland & Hisdal (2008), Eslamian et al. (2010) or Rezaeianzadeh et al. (2013). Some applications to flood inundation modelling have also been investigated (Pender & Liu, 2011), although these are scarce. The aforementioned studies have shown that these computationally less demanding approaches can result in comparable predictions to physically-based models.

In this paper we propose and compare the applica tion of different linear, non-linear and non-parametric regression techniques as surrogate models of the 2D shallow water equations (SWE) applied to flood inun dation mapping in ungauged urban areas. A coastal urban area is used as a test case. The case is specially challenging since the spatial distribution of water depth in the study area depends on the flow discharge in three different tributaries as well as on the tidal level and range.

2 METHODOLOGY

2.1 Case study site

The urban area of Vilagarcia de Arousa (province of Pontevedra, Northwest of Spain) is used as a test case in this study (Fig. 1 and Fig. 2). This coastal town lies at the end of the river Con, which flows into the sea in the Ria de Arousa estuary. It is recurrently affected by flooding and is classified as high potential flood risk area in the Galician Coastal Area Hydrological Plan.

The basin of the river Con has a total area of 24 km^2 and its topography is relatively steep, the maximum height being 640 m.a.s.l. It is divided into three sub catchments, the main one corresponding to the river Con and extending over the central and northern part of the catchment (Q_1 outlet in Fig. 1, with an area of 16.3 km^2). The hydrologic characteristics of the basin, with low concentration time and steep slopes, as well as the tide level play a role in the flooding of the urban area.

2.2 Physically-based model

The physically-based model used in this work com prises a hydraulic model of the urban area ofVilagarcia deArousa (Fig. 2) and a hydrological model of the river Con catchment (Fig. 1). The hydrological model was used to generate the input boundary conditions for the hydraulic model, which subsequently calculated the flow field in the urban area. Historical rainfall events were used to generate the calibration and validation dataset required for the regression models. A brief

Figure 2. Example of maximum water depth field pre dicted by the hydraulic model in the urban area of Vilagarcia de Arousa. Location of control points (1–20) and open boundaries (Q 1 , Q 2 , Q 3 and sea boundary).

triangular elements, with element sizes ranging from 1 to 10 m. The topography was defined from a 1 m res olution gridded LIDAR-based digital surface model. The resulting DEM was enhanced by integrating the geometry of structures and buildings The latter were represented as void areas, i.e., ineffective flow areas. Bed friction was computed with Manning's for mula. Typical values were selected for the Manning roughness coefficient: 0.025 s·m −1/3 for the river bed, 0.032 s·m −1/3 for urban vegetated areas and 0.020 s·m −1/3 for concrete areas. Hydrographs pro vided by the hydrological model were imposed at the three upstream open boundaries: Q 1 , Q 2 and Q 3 in Fig.2. At the sea boundary, the water surface elevation time series was fixed.

The maximum water depth computed by the model

at 20 control points constitutes the output data for each

model run (Fig. 2).

2.2.3 Generation of the dataset for the regression models

A total of 100 model simulations were run to gen

erate the calibration and validation datasets for the

regression techniques. Each model run is defined by a

rainfall time series, a water elevation time series (tide)

and two constant parameters, the initial abstraction and

the potential infiltration in the hydrological model. Table 1. Predictor variables considered in each regression model. Model Model type Input variables 1 LR Q 1–10 2 LR Q 1–10 , Z s 3 LR Q 1–10 , Q 2–10 , Q 3–10 , Z s 4 LS-SVM Q 1–10 5 LS-SVM Q 1–10 , Z s 6 LS-SVM Q 1–10 , Q 2–10 , Q 3–10 , Z s Historical rainfall data recorded at Coron meteorological station were used.The 15 events that resulted in higher rainfall daily amounts in the period 2000–2015 were selected. Initial abstraction and potential infiltration values in the range 0–20 mm and 0–5 mm/h, respectively, were considered. Four different water surface elevation time series at the sea boundary were defined, which correspond to spring tide or neap tide (3 m vs. 1.8 m tide range) and starting either in ebb tide or rising tide (6 hours offset) In order to split the input dataset into validation and calibration parts, k-means clustering analysis was performed (MacQueen, 1967). A total of 25 runs that cover the range of input conditions were selected as calibration data. The remaining 75 runs constitute the validation data. 2.3 Regression models The aim of regression models is to assess whether a dependent variable can be predicted from a set of independent (or predictor) variables. In this case, the dependent variable is the maximum water depth computed by the hydraulic model at the selected control points. The predictor variables were derived from discharge and tide data prescribed at the boundaries of the hydraulic model domain. In the case of the hydrographs, the maximum discharge rate exceeded for 10 minutes was selected as predictor variable (named Q 1–10 ,

Q_{2-10} , Q_{3-10} for the three upstream boundaries, respectively). In the case of the sea water level time series, the maximum value was extracted (named Z_s). Different types of input vectors were considered to develop the regression models (Table 1). A single predictor variable was assessed in models 1 and 4, two predictor variables in models 2 and 5 and a combination of 4 parameters in models 3 and 6. Two types of models were tested: standard linear regression models (LR) and least-squares support vector regression models (LS-SVM).They were developed using Matlab software (version 2011a) and the toolbox StatLSSVM (De Brabanter et al, 2013). Both models are briefly described below. 2.3.1 Standard multiple linear regression The general form of a linear regression model (LR) for n independent variables is given by:

Where Y is the response variable, β_0 , β_1 , β_2 , …, β_n are the regression coefficients, e is the error and X_1 , X_2 , …, X_n are the independent variables. Based on least squares criterion, the regression coefficients are estimated by minimizing the sum of the squares of the vertical deviations of each data point to the best-fitting line.

2.3.2 Least Squares Support Vector regression

Least squares support vector regression models (LS SVM) (Suykens at al., 2002) are derived from the original support vector model (Vapnik 1998), a kernel based learning method for linear and non-linear clas sification problems. The methodology of LS-SVM for regression is briefly described below.

An input vector X is mapped into a high dimensional feature space ψ through a non-linear function φ. In this space, one considers a class of linear functions defined as:

Given the model class $F_{n,\psi}$, a regularization param

eter $\gamma > 0$, and the following training data set:

the LS-SVM for regression is formulated as follows:

This is solved by using Lagrange multipliers $\alpha_i \in R$,

called support vectors. By using Mercer's condition

(further details in De Brabanter et al., 2013), the

resulting LS-SVM model is given by:

Where K is the kernel function, being a Gaussian type

kernel used in this work.

2.3.3 Model performance evaluation

The performance of each of the regression models was

evaluated using the mean absolute error (MAE) and the

coefficient of determination (R^2), defined as follows:

where Y_i is the water depth predicted by the regres

sion model, $\hat{Y}_i$ is the water depth computed by the physically-based model and $\hat{Y}_m$ is the average water depth computed by the physically-based model. These performance measures were evaluated for each model both on a point-by-point basis and globally for all points. Calibration and validation datasets were considered separately. 3 RESULTS AND DISCUSSION The performances of the regression models in terms of MAE and R^2 for calibration and validation phases are shown in Table 2. These are global results for all the control points, hence 20 different regression models are being evaluated in each case. Higher prediction efficiencies are obtained with LS-SVM models, compared with the LR ones. The global MAE is reduced to more than half, being in the order of 0.08 m and 0.03 m in validation mode for LR and LS-SVM models, respectively. A comparison between the water depths computed by the physically-based model and the regression models 3 and 6 is shown in Fig. 3 and Fig. 4. The regression models are forced with the same input vector, the only difference being the model type (LR in Fig. 3 and LS-SVM in Fig. 4). In both cases, a similar model performance is observed for the whole range of water Table 2. Global performance of the regression models for calibration and validation phases. R 2 MAE (m) Model Calibration Validation Calibration Validation 1 0.086 0.089 0.975 0.969 2 0.080 0.077 0.978 0.973 3 0.072 0.074 0.980 0.974 4 0.014 0.026 0.999 0.995 5 0.010 0.027 1.000 0.994 6 0.008 0.026 1.000 0.994 Figure 3. Scatter plot of model 3 (LR type) for the validation phase. Each color represents a different control point.

Figure 4. Scatter plot of model 6 (LS-SVM type) for the

validation phase. Each color represents a different control

point.

Table 3. Number of control points in which MAE is below

5% of the computed water depth range and R 2 is above 0.80,

in validation mode. Number of points (/20)

Model MAE <5%▮Y^ R 2 > 0.80

1 4 13

2 8 15

3 9 15

4 16 19

5 17 19

6 18 19

depth magnitudes. However, the use of the LS-SVM

approach reduces the scatter of the points about the

1:1 line.

If analyzed on a point by point basis, similar obser

vations can be made. LS-SVM models show a superior

performance in all points (Fig. 5), with lower MAE and higher R^2. In nearly all points, the MAE represents less than 5% of the water depth variation computed by the physically-based model (Table 3). Likewise, R^2 is above 0.8 in 19 of the 20 control points for models 4, 5 and 6 (Table 3). On the contrary, MAE with LR models is in the range of 2.3–17.6 % of the water depth variations, with less than half of the control points below 5 %.

Both in LR and LS-SVM models, the inclusion of additional input variables (Z_s in models 2 and 5, Q_{2-10} and Q_{3-10} in models 3 and 6) results in an increased performance, globally and pointwise. However, this increase is not significant. When considering all the points as a whole, variations in MAE are below 1 cm (Table 2). Pointwise, the effect is also limited, in particular with the LS-SVM approach (Table 3). This could be explained by the correlation between the selected predictor variables, in the case of the three input discharges. In the case of the tide, the extraction

Figure 5. Values of R^2 and MAE obtained in each point with models 3 (LR) and 6 (LS-SVM), validation phase.

of a different variable from the water elevation time series could potentially provide better results, although further work is needed to clarify this point. Overall, the regression model number 6, based on LS-SVM, shows the best agreement with the water depth computed with the physically-based model. Nonetheless, the LS-SVM models with fewer input variables (models 4 and 5) produce as well satisfactory results.

4 CONCLUSIONS

In this study, we propose and compare the application of different regression techniques as surrogate models of the 2D shallow water equations. Both linear and non-linear regression models,

with different predictor variables, are developed to compute the water depth field in a coastal urban area. Regression based on least-squares support vector machines resulted in better water depth estimates than standard linear regression, in terms of MAE and R^2. In this particular case, the inclusion of tide level data and three tributaries discharge data as predictor variables can slightly improve model performance. In this way, mean absolute errors represented less than 5.5% of the water depth variation computed by the SWE model at all control points. Due to its simplicity, reduced computation times and good performance, this approach offers great potential for applications such as real time forecasting or uncertainty propagation in a Monte Carlo context.
ACKNOWLEDGEMENTS This work was financially supported by the Spanish Ministry of Economy and Competitiveness (Ministerio de Economía y Competitividad) within the project “CAPRI: Probabilistic flood prediction with high resolution hydrologic models from radar rainfall estimates” (reference CGL2013-46245-R). Maria Bermudez gratefully acknowledges financial support from the Spanish Regional Government of Galicia, Plan I2C (Postdoctoral grant reference ED481B 2014/156-0).

Anctil, F. & Rat,A. 2005. Evaluation of neural network streamflow forecasting on 47 watersheds. J. Hydrol. Eng. 10: 85–88.

Bhagwat, P.P.& Maity, R. 2013. Hydroclimatic streamflow prediction using Least Squares-Support Vector Regression. ISH Journal of Hydraulic Engineering 19(3): 320–328.

Bladé, E., Cea, L., Corestein, G., Escolano, E., Puertas, J., Vázquez-Cendón, M.E., Dolz, J., Coll, A. 2014. Iber: herramienta de simulación numérica del flujo en ríos. Revista

Internacional de Métodos Numéricos para Cálculo y Diseño en Ingeniería, 30 (1): 1-10.

Cea, L. & Blade, E. 2015. A simple and efficient unstructured finite volume scheme for solving the shallow water equations in overland flow applications. Water Resources Research 51(7): 5464-5486.

Cea, L., Legout, C., Darboux, F., Esteves, M., & Nord, G. 2014. Experimental validation of a 2D overland flow model using high resolution water depth and velocity data. Journal of Hydrology, 513: 142-153.

Cea, L. & Vazquez-Cendon, M.E., 2010. Unstructured finite volume discretization of two-dimensional depth-averaged shallow water equations with porosity. Int. J. Num. Meth. Fluid 63: 903-930.

Chang, L.C., Shen, H.Y., Wang, Y.F., Huang, J.Y. and Lin, Y.T. 2010. Clustering-based hybrid inundation model for forecasting flood inundation depths. Journal of Hydrology 385: 257-268.

Chau, K.W., Wu, C.L., Li, Y.S. 2005. Comparison of Several Flood Forecasting Models in Yangtze River. Journal of Hydrologic Engineering 10.1061/(ASCE) 1084-0699 (2005)10:6(485), 485-491.

Chen, A.S., Evans, B., Djordjevic, S., Savic, D.A. 2012. A coarse-grid approach to representing building blockage effects in 2D urban flood modelling. Journal of Hydrology

426-427: 1-16.

Chen, S.M., Wang, Y.M. and Tsou, I. 2013. Using artificial neural network approach for modelling rainfall-runoff due to typhoon. J. Earth Syst. Sci. 122 (2): 399-405.

Chiang, Y., Chang, F. 2009. Integrating hydrometeorologi cal information for rainfall-runoff modelling by artificial neural networks. Hydrological processes 23: 1650-1659.

Dawson, C.W., Abrahart, R.J., Shamseldin, A.Y. and Wilby, R.L, 2006. Flood estimation at ungauged sites using artificial neural networks. Journal of Hydrology, 319: 391-409.

De Brabanter, J.A.K. Suykens & B. De Moor. 2013. Nonpara metric regression via StatLSSVM, Journal of Statistical Software, 55(2): 1-21. Engeland, K. & Hisdal, H. 2009. A Comparison of Low Flow Estimates in Ungauged Catchments Using Regional Regression and the HBV-Model. Water Resour Manage 23: 2567-2586. Eslamian, S., Ghasemizadeh, M., Biabanaki, M.,Talebizadeh, M. 2010. A Principal Component Regression Model for Estimating Low Flow Index. Water Resour Manage 24: 2553-2566. Fewtrell, T.J., Bates, P.D., Horrit, M., Hunter, N.M. 2008. Evaluating the effect of scale in flood inundation modelling in urban environments. Hydrol Process 22, 5107-5118. Guinot, V., 2012. Multiple porosity shallow water models for macroscopic modelling of urban floods. Adv. Water Resour. 37: 40-72. Hunter, N.M., Bates, P.D., Neelz, S., Pender, G., Villanueva, I., Wright, N.G., Liang, D., Falconer, R.A., Lin, B., Waller, S., Crossley, A.J. and Mason, D. 2008. Benchmarking 2D hydraulic models for urban flood simulations. Proceedings of the Institution of Civil Engineers: Water Management, 161(1): 13-30. Li, X., Lu, H., Horton, R.,An, T.,Yu, T. 2014. Real-time flood forecast using the coupling support vector machine and data assimilation method. Journal of Hydroinformatics 16(5): 973-988. MacQueen, J.B. 1967. Some methods for classification and analysis of multivariate observations. Proceedings of the Fifth Symposium on Mathematical Statistics and Probability, Volume I, Statistics. University of California Press. Pender, G. & Liu, Y. 2011. Rapid Flood Inundation Modelling: Meta-Modelling of 2D Hydrodynamic Model Using Artificial Intelligence Techniques (Support Vector Regression, Modified Linear Interpolation and Cellular Automata). FRMRC flood risk management consortium. FRMRC Research Report WP1.3 . Rezaeianzadeh, M., Tabari, H., Arabi Yazdi, A., Isik, S., Kalin, L. 2014. Flood flow forecasting usingANN,ANFIS and regression models. Neural Comput & Applic 25(1): 25–37. Sanders, B.F., Schubert, J.E., Gallegos, H.A. 2008. Integral formulation of shallow water equations with anisotropic porosity for urban flood modeling. Journal of Hydrology 362: 19–38. Schubert, J.E., Sanders, B.F., 2012. Building treatments for urban flood inundation models and implications for predictive skill and modelling efficiency. Adv. Water Resources 41: 49–64. Suykens, J.A.K., Van Gestel, T., De Brabanter, J., De Moor, B., Vandewalle, J. 2002. Least Squares Support Vector Machines. World Scientific, Singapore. Vapnik VN. 1998. Statistical Learning Theory. Wiley Yu, D. & Lane, S.N. 2006. Urban fluvial flood modelling using a two-dimensional diffusion-wave treatment, part 1: mesh resolution effects. Hydrological Processes 20(7): 1541–1565. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Impacts of urban expansion on future flood damage: A case study

in the River Meuse basin, Belgium

A. Mustafa

ArGEnCo Department, University of Liege, Belgium

LEMA Research Group

M. Bruwier

ArGEnCo Department, University of Liege, Belgium

Hydraulics in Environmental and Civil Engineering (HECE) Research Group

J. Teller

ArGEnCo Department, University of Liege, Belgium

LEMA Research Group

P. Archambeau, S. Erpicum, M. Pirotton & B. Dewals

ArGEnCo Department, University of Liege, Belgium

Hydraulics in Environmental and Civil Engineering (HECE) Research Group

ABSTRACT: Climate change and urban development are key factors influencing future flood damage. This

paper evaluates the sensitivity of future flood damage to a number of urban expansion scenarios for the river

Meuse in Belgium. Based on these scenarios, the impact of urban development on flood damage is assessed.

The study uses the multinomial logistic regression model, which enables visualization of the consequences

of different urban densities expansion. Land registry data for the years 1990, 2000 and 2010 were used to

define four classes (non-urban, low-density, medium-density and high-density urban). Besides, several socio

economic, geographic and political driving forces for urban development were operationalized to create maps

of urbanization likelihood. These maps were then used to predict future urban scenarios for 2020 and 2030. The

flood damage estimation was performed for urban lands by overlaying the inundation map for a specific flood

discharge with the different urban expansion scenarios and by using a damage curve and specific prices.

1 INTRODUCTION

It is widely accepted that the magnitude and frequency

of river flooding is currently increasing in North

west Europe (Hannaford and Marsh, 2008; Moel and

Aerts, 2010; Robson, 2002). Besides climate change,

increases in flood damage can mainly be attributed to urban development in flood-prone areas (Moel and Aerts, 2010). Therefore, there is a high policy interest in clarifying the links between urban development and flood damage.

This work addresses how flood damage is influenced by urban expansion within the flood-prone areas along river Meuse in the Walloon region (Belgium). First, an urban expansion model is developed to derive expansion scenarios for 2020 and 2030. Second, the corresponding flood damage is estimated for different urban expansion scenarios.

Most existing urban expansion models are based on a regular grid composed of square cells of dimensions between 30 × 30 m^2 and 300 × 300 m^2 (e.g. Guan et al., 2011; Hu and Lo, 2007; Jantz et al., 2003; Liu et al., 2008; Mustafa et al., 2014; Rienow and Goetzke, 2015). These models address urban expansion as a binary process, through the identification of urban versus non-urban land-uses. Most urban cells at these dimensions comprise a mix of different land-uses. This causes erroneous estimations of urban expansion patterns. The present paper proposes an urban expansion model that enables modelling three urban classes: low-density urban, medium-density urban and high-density urban. The model is based on a multinomial logistic regression (MLR) model. Belgian land registry data (CAD) were used to identify urban densities. Urban land-use maps have been prepared for years 1990, 2000 and 2010 for the Walloon region based on CAD data. Next, the MLR model has been applied to correlate the observed urban expansion patterns for different urban densities with a number of indicators related to distances, topological, neighbourhood, socioeconomic factors and land-use policies. Finally, the MLR outcomes were used to derive urban expansion scenarios for 2020 and 2030. Relative operating

Figure 1. Study area.

characteristic (ROC) method validates the MLR out comes. This paper focuses on assessing the transitions from non-urban land-use into one of the urban density classes (and not on urban densification).

2 MATERIAL AND METHODS

2.1 Study area

River Meuse is one of the major European rivers. It originates in France and flows through Belgium and the Netherlands before reaching the North Sea. It has a total length of 905 km with 152 km in the Walloon region. The flood-prone areas along river Meuse in the Walloon region define the extent of the present analysis (Figure 1).

2.2 Urban expansion model

The urban expansion modelling consists of two main steps: (i) development probability maps for the three urban classes (low-density, medium-density and high density urban) versus non-urban class; and (ii) sce narios of future urban patterns for the years 2020 and 2030.

The MLR was used to develop probability maps.

The dependent variables for the MLR model were defined as the changes from a non-urban class to one of the three urban density classes using CAD. CAD is a vector dataset representing buildings in two dimen

sions as polygons. Each building comes with different attributes, from which the construction date is the most important one in this study. Using the construction date, three urban land-use raster-grids were gener ated for years 1990, 2000 and 2010. CAD vector data were rasterized at a very fine resolution (2 × 2 m 2). The rasterized cells were then aggregated to obtain a 50 × 50 m 2 raster-grid. Thus, each aggregated cell has a density value that exhibits a number of the raster ized 2 × 2 m 2 cells. The magnitude of density value was then used to represent four classes: non-urban (class 0), low-density urban (class 1), medium-density urban (class 2) and high-density urban (class 3). The geometrical interval classification method was used to set the thresholds defining the urban land-use classes
Table 1. List of selected urbanization driving forces. Driver Name Unit X1 Elevation Meter X2 Slope Percent X3 Distance to roads (1st category) Meter X4 Distance to roads (2nd category) Meter X5 Distance to roads (3rd category) Meter X6 Distance to roads (4th category) Meter X7 Distance to railway stations Meter X8 Distance to large cities Meter X9 Distance to medium cities Meter X10 Number of class1 cells within a Number 5 × 5 window X11 Number of class2 cells within a Number 5 × 5 window X12 Number of class3 cells within a Number 5 × 5 window X13 Population density inh/km 2 X14 Employment rate Percent X15 Zoning Binary (density values for class 1: 4–9.12%, class 2: 9.12-28% and class 3: 28–100%). The independent urbanization driving forces, X, were selected based on expert knowledge of the study area as well from literature review (Cammerer et al., 2013; Mustafa et al., 2014). These drivers can be grouped into five sets: (i) distance-related factors, (ii) topological factors, (iii) neighbourhood factors, (iv) socio-economic factors and (v) land-use policies. Table 1 summarizes the complete list of selected urbanization

driving forces. A zoning map (land-use policy) was developed by discriminating between the zones where urban development is permitted and those were it is not in the regional development plan. The driving forces are measured in different units and therefore we standardized all continuous drivers. Since some of the selected driving forces measure the same phenomena, strong collinearities would cause erroneous estimations of the MLR’s parameters. Consequently, a multicollinearity test was examined in the initial stage using the variance inflation factors (VIF). Montgomery and Runger (2003) recommended that the VIFs should not exceed 4. The VIF test results for all standardized driving forces suggest that the variables elevation (X1) and slope (X2) measure the same phenomena as well as population density (X13) and employment rate (X14). Therefore, the elevation and employment rate variables were suppressed. All remaining X variables show an acceptable degree of multicollinearity and can therefore be introduced in the MLR. Both dependent and independent variables may exhibit spatial autocorrelation, which biases the results of the regression analysis (Overmars et al., 2003). This issue can be addressed through a data sampling approach (Cammerer et al., 2013; Rienow and Goetzke, 2015). For the model calibration, 45,000 cells were randomly selected.

The general form of the MLR can be represented as:

where $g_{cl,n}$ is the logit function of class cl_n versus the

reference non-urban class cl_0 , α is the intercept term

for class cl_n versus the reference class cl_0 and β is

the coefficient vector. Thus, the probability $P(i,cl_n)$ of

change from class 0 to class n in a cell i is given by:

2.3 Inundation modelling

The inundation extent and water depths were com

puted all along the Walloon part of river Meuse for

Q_{100} + 30%, the 100-year flood discharge plus 30%

(4095 m^3/s in the city of Liege).The computation of the

inundation characteristics does not account for land

use changes. The simulations were performed with a

5×5 m 2 raster-grid with the hydraulic model Wolf 2D. Wolf 2D is a two-dimensional hydraulic model which solves the complete shallow-water equations based on a conservative finite volume scheme using a flux vector splitting technique (Erpicum et al., 2010). This model was extensively used for inundation mod elling in flood damage analysis (Beckers et al., 2013; Bruwier et al., 2015).

2.4 Flood damage assessment

The flood damage related to a land-use scenario was evaluated by combining the corresponding land use map with the inundation map and by using a stage-damage function for urban areas and specific prices (mobile and immobile). Urban areas are indeed associated to most flood damages along the river Meuse.

Stage-damage functions were used to determine the relative damage corresponding to an inundation depth. The Flood Loss Estimation Model (FLEMO) defines stage-damage functions for residential, industrial and commercial land plots (Kreibich et al., 2010). In this study, all urban land-use categories are aggregated within a single urban category. Since the residential category contributes most to flood damage in the Wal loon part of river Meuse, we decided to assign the residential stage-damage to the whole urban category.

The same function is used for both mobile and immo

bile prices of urban areas, consistently with the study

of Beckers et al. (2013).

The specific prices were assessed based on Beck

ers et al. (2013), who introduced specific mobile and

immobile prices derived from ATKIS data (Muller

2000) and adapted from Germany to the Walloon

region. In this paper, the prices for each urban class was

computed as the product of the average density over all

urban cells of this specific urban density class times

the price per m 2 of building. This latter price was cali

brated to correspond to the flood damage computed for 2010 with the one computed by Beckers et al. (2013) for 2009 (immobile and mobile price were calibrated respectively to 1500 €m −2 and 430 €m −2). For each land-use cell (50 × 50 m) labelled as urban (class 1, class 2 or class 3), we extracted the cells from the inundation map (5 × 5 m 2) which are included within this land-use cell. After excluding the inundation cells lying in the main riverbed of the river Meuse, the stagedamage function was used to determine the relative damage associated to each urban cell. The absolute damage related to the urban cell was obtained by multiplying the sum of relative damage over the urban cell by the specific mobile and immobile prices. Finally, absolute damages were summed over all urban cells to obtain the total damage. 2.5 Influence of the number of urban classes on the flood damage As presented in section 2.2, the urban expansion model distinguishes three urban density classes. The evaluation of the flood damage is dependent on the number of classes since the cost-damage function are a nonlinear function of the water depth while the specific prices for each urban class are equal to the product of the price per m 2 of building times the average density of this class. The influence of the number of urban classes on the flood damage is quantified for 2010 by computing the flood damage with one to eight urban classes.

3 RESULTS AND DISCUSSION 3.1 Future land-use maps Figure 2 shows the three urban density classes of the 2010 urban land-use map. High-density urban lands are concentrated in the existing urban centres. Medium-density lands tend to be located in suburbs around cities and low-density lands are likely to be found in rural and remote locations. The MLR's outcomes are probability maps of urbanization for each class based on vectors of regression coefficients β and intercepts α. Table 2 lists the MLR's results. All driving forces are statistically significant on one or more urban classes. The impact of different drivers varies along with urban density. However, the zoning status (X15) is by far the most important driving force for all urban density classes. The influence of zoning reveals a decreasing trend from class 1 (low-density) to class 2 and class 3 respectively. It is quite understandable that zoning has a larger impact in low-density areas where planned urban zones stock is large, far from existing urban centres. According to the slope driver (X2), low-density urban expansion tends to occur on relatively hilly terrains, whereas medium and high-density urban areas are more developed in flat areas. Distances to major roads (1st and 2nd categories, X3 and X4) have a

Figure 2. Urban density classes for 2010.

noticeable impact on the development of high den

sity areas (class 3). In contrast, these road categories

show no significant contribution to the low-density

urban developments. The results also reveal that the

impact of distance to local roads (X6) is generally

decreasing with increasing urban densities. Distance

to railway stations (X7) attracts high-density urban

developments and repels low-density urban develop

ments. The number of cells of different urban densities

classes within a 5 × 5 window (X10, X11 and X12)

demonstrates a significant contribution to all urban

density classes except for X10 to high-density urban class. The low-density class shows a strong relation ship with the increasing number of existing low-urban density cells within a neighbourhood of 5 × 5 win dow size (X10). The probability of medium-density urban developments is increased with an increasing number of existing low, high and medium-density urban cells in the neighbourhood. Contrarily, the prob ability of high-density expansion is mainly increased by a higher number of existing high-urban density cells within a neighbourhood of 5 × 5 window size (X12). The contribution of population density (X13) to urban expansion is statistically significant only for mediumand high-density urban expansions. Logi cally, the medium and highdensity developments are likely to occur in densely populated areas. In contrast, low-density developments are often characterized by a lower population density as they usually correspond to the replacement of farmland, forest and open space by single-family homes on relatively large lots.

The ROC values of the probability maps are 0.94, 0.93 and 0.88 for classes 1, 2 and 3 respectively. ROC values higher than 0.70 are considered as a reasonable fit (Cammerer et al., 2013; Hosmer and Lemeshow, 2004; Poelmans, 2010). Hence, these probability maps can be used for reliable predictions of the near future urban expansion patterns.

The developed probability maps were used to

generate future urban scenarios for 2020 and 2030 by

(i) quantifying the necessary area for future expansion

for each urban density class and (ii) selecting the cells

of the probability maps with the highest values for

each class until the required areas are met. This gener

ates urban expansion maps for 2020 and 2030 years. Table 2. Intercept α and coefficients β of the MLR model (class 0 is the reference class). Driver Class1 Class2 Class3 Intercept -2.869 -2.979 -3.381 X1 N.I.** N.I. N.I. X2 0.036 -0.148* -0.141* X3 -0.038 -0.125* -0.534* X4 -0.004 -0.112* -0.350* X5 -0.137* -0.160* -0.238* X6 -0.356* -0.306* -0.160* X7 0.097* 0.001 -0.174* X8 0.006 0.053* 0.122* X9 -0.054* -0.104* -0.069* X10 0.391* 0.305* -0.024 X11 0.153* 0.151* 0.036* X12 0.091* 0.214* 0.460* X13 0.000 0.147* 0.138* X14 N.I. N.I. N.I. X15 3.317* 2.930* 2.692* * Indicates significance at P ≥ 0.05 ** Not included Figure 3. Coefficients β of the MLR model. Waterbodies, that are defined using zoning plan, are considered as exclusion zones. The quantity of new urban land can be estimated by several means, including the Markov chain model (e.g. Sang et al., 2011;Yang et al., 2014), linear extrapolation of past trends (Mustafa et al., 2014; Poelmans et al., 2010) and/or based on socioeconomic factors (e.g. White and Engelen, 2000). Based on linear extrapolations of the real urban expansion between 1990, 2000 and 2010 for Wallonia, three urban expansion scenarios were proposed for 2020 and 2030: low-, mediumand high-demand. These scenarios were derived by extrapolating respectively the trends between 2000 and 2010, 1990 and 2010, and 1990 and 2000 (Figure 4). 3.2 Influence of the number of urban classes on the flood damage Figure 5 represents the computed flood damage for the flood discharge Q 100 + 30%, as a function of the number of considered urban classes (1 to 8). The computed damage is underestimated by 22% when only

Figure 4. Urban scenarios of 2020 and 2030.

Figure 5. Flood damage as a function of the number of urban

classes.

one urban class is used compared to the damage com puted for eight urban classes. Once the number of urban classes is higher than two, the computed flood damage becomes slightly affected by the number of urban classes, with variations lower than 4%. Hence, the use of three urban classes to generate the future land-use maps is considered acceptable in terms of independence of the estimated flood damage to the number of urban classes.

3.3 Impact of urbanization on future flood damage

Figure 6 shows the relative surface expansion of each land-use class for 2020 and 2030 compared to the reference (2010). The non-urban area decreases from 2010 to 2030 due to the urbanization process. Gener ally, the relative surface expansion is higher for classes with a lower density (except for the medium-density Figure 6. Relative surface expansion of each land-use class for future time-horizons 2020 and 2030 compared to 2010. class for the low and medium expansion scenarios of 2020). The sensitivity to the expansion scenario is higher for 2030 than for 2020 with a range of variation of the total surface expansion respectively between −3% and 12% and −5.5% and 29%. In Figure 7, the ranges of increases in flood damage are between 14% and 21% for 2020 and between 23% and 38% for 2030. Compared to the repartition of the flood damage between the different urban classes for 2010 (respectively 80, 470 and 1,305 millions euros for the low, medium and high-density classes), the highest increase in relative flood damage occurs for the medium-density class, while the rise in flood damage for the high-density class is slightly higher than for the low-density class. The increase in the total flood damage is mainly related to the high-density class (for approximately 60%) while the contribution of the low-density class is low (lower than 4%). 4 CONCLUSION This paper employs the multinomial logistic regression (MLR) model to examine drivers of urban expansion in the Walloon region (Belgium) and to forecast 2020 and 2030 urban expansions using Belgian land registry data (CAD). Four classes (non-urban, low-density, mediumdensity and high-density urban) were defined as dependent variables for the MLR. Several variables

Figure 7. Relative increase in the flood damage due to the future urbanization for 2020 and 2030 compared to the one computed for 2010.

were selected and introduced in the MLR model as independent driving forces. It was found that all the independent variables have an impact on urban expansion in the Walloon region, but their relative importance varies with the urban class. It can also be concluded that zoning is by far the most important determinant of urban expansion process for this case study.A validation of the MLR showed that the model’s outcomes allow predicting future urban expansion patterns with a relatively high explanatory power. Based on linear extrapolation of urban expansion between 1990, 2000 and 2010, three expansion sce narios are proposed to simulate 2020 and 2030 urban patterns (low, medium and high demand scenarios). Most importantly, this work highlights that the use of several urban density classes to evaluate the flood damage enables to improve the computation since the price of each class can be better addressed. However, the benefit of using an increasing number of urban classes drops beyond three urban classes.

Based on the future urban expansion maps, the impact of future urbanization on future flood dam age related to the Q 100 + 30% discharge was assessed. While the urban expansion is higher for the low density class, the flood damage increases more for the medium-density class. The increases in flood damage compared with 2010 are around 14–21% for 2020 and

non-linear cellular automata. Ecol. Model. 211, 169–181. doi:10.1016/j.ecolmodel.2007.08.024

Moel, H. de, Aerts, J.C.J.H., 2010. Effect of uncertainty in land use, damage models and inundation depth on flood damage estimates. Nat. Hazards 58, 407–425. doi:10.1007/s11069-010-9675-6

Montgomery, D.C., Runger, G.C., 2003. Applied Statistics and Probability for Engineers, Fourth. ed. John Wiley & Sons, New York.

Muller, W., 2000. ATKIS®data base revision and gener ation of digital topographic base maps. Int. Arch. of Photogramm. Remote Sens., 33, 710–717.

Mustafa, A., Saadi, I., Cools, M., Teller, J., 2014. Measuring the Effect of Stochastic Perturbation Com ponent in Cellular Automata Urban Growth Model. Procedia Environ. Sci., 12th International Conference on Design and Decision Support Systems in Architec ture and Urban Planning, DDSS 2014 22, 156–168.

doi:10.1016/j.proenv.2014.11.016

Overmars, K.P., de Koning, G.H.J., Veldkamp, A., 2003.

Spatial autocorrelation in multi-scale land use mod

els. Ecol. Model. 164, 257–270. doi:10.1016/S0304

3800(03)00070-X

Poelmans, L., 2010. Modelling urban expansion and its

hydrological impacts (Unpublished PhD dissertation).

Katholieke Universiteit Leuven, Leuven. Poelmans, L., Van Rompaey, A., Batelaan, O., 2010. Coupling urban expansion models and hydrological models: How important are spatial patterns? Land Use Policy 27, 965–975. doi:10.1016/j.landusepol.2009.12.010 Rienow, A., Goetzke, R., 2015. Supporting SLEUTH – Enhancing a cellular automaton with support vector machines for urban growth modeling. Comput. Environ. Urban Syst. 49, 66–81. doi:10.1016/j.compenvurbsys. 2014.05.001 Robson, A.J., 2002. Evidence for trends in UK flooding. Philos. Transact. A Math. Phys. Eng. Sci. 360, 1327–1343. doi:10.1098/rsta.2002.1003 Sang, L., Zhang, C.,Yang, J., Zhu, D.,Yun, W., 2011. Simulation of land use spatial pattern of towns and villages based on CA–Markov model. Math. Comput. Model., Mathematical and Computer Modeling in agriculture (CCTA 2010) 54, 938–943. doi:10.1016/j.mcm.2010.11.019 White, R., Engelen, G., 2000. High-resolution integrated modelling of the spatial dynamics of urban and regional systems. Comput. Environ. Urban Syst. 24, 383–400. doi:10.1016/S0198-9715(00)00012-0 Yang, X., Zheng, X.-Q., Chen, R., 2014. A land use change model: Integrating landscape pattern indexes and Markov-CA. Ecol. Model. 283, 1–7. doi:10.1016/ j.ecolmodel.2014.03.011
Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Uncertainty assessment of river water levels on energy head loss through hydraulic control structures

P. Meert

Hydraulics Section, Department of Civil Engineering, KU Leuven, Leuven, Belgium

F. Pereira

Flanders Hydraulics Research, Antwerp, Belgium

P. Willems

Hydraulics Section, Department of Civil Engineering, KU Leuven, Leuven, Belgium

Department of Hydrology and Hydraulic Engineering, Vrije Universiteit Brussel, Brussels, Belgium

ABSTRACT: Detailed hydrodynamic models that solve the full de Saint-Venant equations provide a powerful

tool to investigate the impact of man-made interactions, like sluices and weirs, in the river system. To model

these hydraulic structures in hydrodynamic models, two types of parameters have to be provided: the geometric

properties of the structures and a number of coefficients that are used in the discharge calculation schemes.

The latter are quite hard to quantify and default values are often used. In this paper, a sensitivity analysis was

performed to investigate how the simulation results are affected by these parameter values. In order to reduce

calculation times, a conceptual reservoir-type model was set-up. This surrogate conceptual model was calibrated

to the full hydrodynamic model of the river and shows only minor loss of accuracy, compared to the detailed

model. The parameter values of six new adjustable hydraulic structures in the river Dender, in Belgium, and their

influence on the water levels in the river and the surrounding floodplains were investigated.

1 INTRODUCTION

The impact of adjustable hydraulic structures and their operational gate regulation on river or reservoir water levels is mostly studied with one-dimensional hydro dynamic models, like InfoWorks RS and MIKE11 (e.g. Ngo et al., 2008). The implementation of hydraulic structures in such models requires a vast amount of information. The geometric properties of the struc ture (e.g. width, sill level, gate height) are quite easy to obtain. Other parameters such as head loss factors or discharge coefficients are more difficult to assess. Therefore, default values are most often used.

Given that modern water management and engi neering is more and more based on model results and that errors and uncertainties in the results of hydrolog ical and hydraulic models often are substantial, it is of great importance that the uncertainties in the model results are taken into account in the decision mak ing processes. Good modelling practice thus not only provides the model results, but also the uncertainty in these results. Quantifying the uncertainties in the model results supplies decision-makers with impor tant additional information in support of their tasks (Willems, 2012).

This paper deals with the investigation of the influ ence of the aforementioned default parameter values on the model simulation results, for the river Dender case in Belgium, based on a hydrodynamic model implemented in

MIKE11 (DHI, 2011). A thorough sensitivity analysis with the detailed hydrodynamic model is very unpractical and time demanding due to the long simulation times. This can be overcome by the use of surrogate conceptual models, that aim to mimic the results of the detailed models, based on a concatenation of reservoir-type model elements. The conceptual models considered in this paper can be regarded as physically-inspired or grey-box models, since their structure is based on a simplified representation of reality (Knight & Shamseldin, 2006). The explicit calculation scheme of these conceptual models allows for short simulation times, making them very suitable for applications that require long-term simulations or a large number of model runs, such as real-time control, uncertainty and sensitivity analyses. The paper first describes the study area and the available data of the river Dender in Belgium. This is followed by a presentation of the methodology used to transform the detailed hydrodynamic model in a lumped conceptual model and a description of the methodology used in the sensitivity analysis. Next, the simulation results of the conceptual model are compared with the results of the detailed model to show the accuracy and the applicability of the conceptual model. Finally, the results of the sensitivity analysis are shown and analyzed.

2 STUDY AREA AND AVAILABLE DATA

The river Dender is part of the international Scheldt basin that flows to the North Sea and is located in the central part of Belgium. The basin has an area of 1384 km ^{2} , whereof 675 km ^{2} is located in the Wal loon Region and the remaining 709 km ^{2} in the Flemish Region. In this area, the river flows in northern direc tion over an approximate length of 45 km towards the city of Dendermonde, where it joins the tidal river Scheldt.

At this moment, the flows and water levels in the river are completely regulated by eight hydraulic struc tures (weirs and sluices, combined with locks). These structures where installed in the second half of the 19th century to regulate the water levels in the river during low flow periods, to allow navigation of trans portation ships, but also to control the flow during storm conditions. The discharge capacity of these out dated structures is, however, insufficient to prevent the river from flooding and thereby inundating the densely populated surrounding areas. The weir in Ger aardsbergen, for example, has an estimated maximum discharge capacity of 45 m 3 /s, which was insufficient to withstand the severe flood of November 2010 with a peak discharge of about 115 m 3 /s. In fact, this maxi mum discharge capacity has an exceedance frequency of twice a year, which often results in inundations upstream of this weir.

To address this problem, the Flemish government has recently decided to renew the hydraulic control structures, to increase the discharge capacity and to ensure their reliability.To do so, the old manually oper ated beam weirs will be replaced by automated gated weirs, which are considerably wider than the old weirs. Furthermore, the old weir at Teralfene (see Figure 1) will be removed, because the water level difference over the weir is relatively small.

A detailed quasi-2D hydrodynamic model of the river Dender in the Flemish region, including the new control structures, was set up in the MIKE11 software of DHI (DHI, 2011). This model contains measured cross-sections approximately every 130 m, as well as all relevant hydraulic structures. Flood plains were implemented at both sides, along the full length of the river, using the quasi-2D modelling approach (Willems, 2013). Simulation results of two flood events (Dec/1999 and Nov/2010) were used for calibrating the conceptual model, while two other events (Jan/1995 and Dec/2002-Jan/2003) were used for validation. A detailed analysis of these calibration and validation results was provided by Meert et al. (2016a).

3 METHODOLOGY

3.1 Conceptual modelling approach

In the conceptual modelling approach, the river net work is modelled in a lumped way: the most important processes are aggregated in space and time. So, instead

Figure 1. Location of the control structures in the river Dender catchment. Green lock symbols indicate structures to be renewed, red symbol the structure to be removed and yellow symbol the recently renovated structure.

of calculating water levels and discharges by solving the full de Saint-Venant equations, values are only calculated in significant and relevant nodes. Therefore, the river network reaches are schematized by a number of mutually interconnected storage reservoirs. The boundaries of these reaches are defined by hydraulic structures such as weirs, sluices, pumps or fixed elements, or by sudden changes in the river geometry that cause important backwater effects. Floodplains can be modelled in a similar way (Wolfs et al., 2015). The discharge rate between two adjacent reservoirs is based calculated with the reservoir routing technique proposed by Fenton (1992). This technique uses a numerical solution for the storage equation that governs the rate of change of reservoir storage volume as a function of inflow and outflow: where S represents storage, t time, I inflow and Q outflow discharge. To solve Equation 1, the outflow has to be related to the storage volume, which is usually done via the dependence of both with the water level. In this paper, a first order decomposition is used: with ∆t the time step. The storage at the current time step is thus estimated, based on storage and discharge values of the previous time step. Hereby, it is assumed that storage

Figure 2. Calculation scheme of the conceptual modelling approach. Full red boxes indicate the calculation steps, dashed

blue boxes refer to calculated values or boundary conditions.

varies only slightly in between two time steps, thereby limiting the time step length. In the next step, the stor age value is transformed into water levels at one or more locations, using calibrated storage water level relations. The calibration of these relations is based on simulation results of the detailed hydrodynamic model. The outflow discharge is calculated with a set of discharge equations that describe the flow through a hydraulic structure, as a function of the water levels at both sides of the structure. In this study, the same cal culation scheme as applied in the MIKE11 software was considered (see section 3.2). This is needed to allow for a detailed sensitivity analysis of the parame ter values. When no hydraulic structure is present, the outflow might be related to the storage in the reservoir or a fictitious structure can be used (e.g. Meert et al., 2016b). All model components and their relative com putation order are presented in Figure 2, and discussed with more details by Wolfs et al. (2015).

3.2 Hydraulic structure modelling

Streamlines in the vicinity of hydraulic structures are subjected to contraction and expansion, due to the sud den change in cross-sectional area.This results in rapid changes to flow velocities and in large-scale turbu lence that dissipates energy as heat.This type of energy loss, near hydraulic structures, is hereafter referred to as form losses. One-dimensional (1D) hydraulic modelling schemes, that solve the 1D form of the de Saint-Venant equations, cannot accurately model these rapid changes and related losses. Therefore, they use special structure equations, where contraction and expansion loss coefficients have to be specified, to account for the energy head loss (Syme, 2001).

This type of losses at hydraulic structures of short length, like the considered overflow structures in the river Dender, are the dominant energy loss compo nents. Energy losses due to friction are small and can be neglected. The head loss during subcritical flow conditions is usually expressed as a function of the dynamic head: with ∆H the energy head loss, ζ the form loss coefficient, V the mean cross-sectional velocity, g the gravitational acceleration, Q the discharge and A the cross-sectional area. In the MIKE11 engine, the total head loss is divided into inflow (or contraction) and outflow (or expansion) components: ζ_1 represents the contraction form loss coefficient and ζ_2 the expansion form loss coefficient. Values for both form loss coefficients can be found in most textbooks on fluid dynamics: ζ_1 typically ranges from 0.0 to 0.7, whereas ζ_2 varies between 0.0 and 1.0 (Syme, 2001). The main factor defining the value of the form loss coefficients is the ratio of the mean current velocity in the river and at the structure. Consider, for example, the case where the approach velocity equals the velocity at the structure. The inflow component of the total head loss should then reduce to zero, because there is no contraction of the stream lines. The MIKE11 engine incorporates this principle, by adjusting the values of contraction and expansion form losses, based on the velocities upand downstream of the structure: with ζ_{in} and ζ_{out} user-defined contraction and expansion form loss coefficients. The subscripts 1, 2 and s refer to the locations, respectively upstream and downstream of the structure and at the structure. MIKE11 distinguishes two types of subcritical flow through hydraulic structures: critical and drowned flow. The former occurs when the water level at the structure is larger than the downstream energy head. In this case, the downstream water level has no influence on the flow and only contraction form losses are considered. The user has the possibility to apply a correction factor α_c to this critical discharge. This critical flow correction factor can be regarded as a discharge coefficient that allows to change the relation between upstream water level and discharge, to

Figure 3. Comparison simulation results for selected nodes in the most upstream reach of the river Dender, for the event of

No-vember 2010: detailed MIKE11 model (blue), original conceptual model (red) and adapted conceptual model (green).

meet measured values. During drowned flow condi

tions the discharge through the structure is affected

by the downstream water level and both form loss components are accounted for.

Overall, there are three parameters influencing the discharge equations, apart from the geometry of the structure and the gate level. The MIKE11 reference manual (DHI, 2011) advices three default values for these coefficients: 0.5, 1.0 and 1.0 for respectively the contraction and expansion form loss coefficient and the critical flow correction factor.

3.3 Rotating water surface profile

As mentioned before, water levels are calculated with storage water level relations that are calibrated to sim ulation results of the detailed model. These usually produce accurate approximations of the water levels. However, changing the parameters of the hydraulic structure at the end of a reach will alter the local (h, Q) relation and influence the backwater effects towards the more upstream located water level nodes.As a con sequence, the curve of the water surface profile in the reach will change, which is currently not accounted for in the conceptual model.

Figure 3 compares the simulation results of the detailed and conceptual models in three nodes: two water level nodes in a reach upstream of the struc ture at Geraardsbergen and the discharge node at that structure. The discharge capacity of the structure was decreased to about 60% of its original capacity, by setting the critical flow correction factor α_c to 0.6. The differences between both model results can be explained as follows: the conceptual model will try to route the same flow through the structure, indepen dent of the discharge capacity of that structure (see Figure 3c). To achieve this, and because the same dis charge calculation scheme is used in both models, the water level just upstream of the structure is also very well approximated (Figure 3b). This upstream water level is, however, higher than in the original model, because of the decreased discharge capacity. To reach this water level, the conceptual model has to increase the stored volume in the reservoir upstream of the structure. This will affect all other water level nodes in the reach (Figure 3a), since they are all linked to this single storage value, and the change in water surface profile shape is not accounted for. Based on this observation, a new model component was introduced. The water level at the structure and the flow through the structure are used to calculate all other water levels, given that they are in all conditions very well approximated. The water level upstream of the structure and the outflow are first calculated with the original model components. All other water level nodes in the reservoir are calculated with a two-dimensional interpolation surface, as a function of the previously calculated water level and outflow discharge. This 2D interpolation surface is calibrated to the simulation results of the detailed model. Note that this is an imitation of the calculation of an M2 water surface profile, without explicitly solving the Bresse equation (e.g. Verhoeven et al., 2003) at every time step. The calibration of these new HQH-relations was performed for the simulation results of two model runs on the Nov/2010 event. The first run was based on the default parameter values, whereas the discharge capacity of all structures was strongly reduced in a second run.This allows to cover a broad range of possible situations in the sensitivity analysis. For both runs the same time series of gate levels were applied. This strategy was chosen to neglect the influence of the gate level regulation schemes, that try to keep the water levels upstream of the structure in a range of 5 cm around a target value. The simulation results of the conceptual model with this new approach are also shown in Figure 3. It is clear that the new model component provides results that are closer to the detailed model. For the water level node shown in Figure 3a the RMSE (see Equation 7) has decreased from 16.8 cm for the original model to 3.5 cm for the new conceptual model, which is an important improvement. 3.4 Sensitivity analysis Helton and Davis (2003) define sensitivity analysis as an attempt to determine how the uncertainty in individual elements of the input affects the uncertainty in the model output. This can be achieved, if the statistics of the input are known (or assumed to be known), by

Table 1. Specifications triangular possibility density func tions used in sensitivity analysis.

Parameter	Lower limit	Upper limit	Mode
ζ_{in}	0.0	0.7	0.5
ζ_{out}	0.0	1.0	1.0
α_c	0.5	1.25	1.0

performing multiple model runs and randomly sampling in the input space. The obtained collection of model outputs then allows one to derive the statistics of the output (Montanari, 2007). In this paper, the Latin Hypercube Sampling technique was chosen, rather than complete random sampling, because the former has proven to produce more stable results for the same number of samples (e.g. Helton & Davis, 2003; McKay et al., 1979).

For all parameters a triangular distribution is assumed, with a width that corresponds to the the oretical range of the parameter and the mode at the default value. These values are listed for each param eter in Table 1. Furthermore, it is assumed that the errors on all parameters are nearly independent, and that the error distributions are propagated separately through the model. This will allow to investigate the sensitivity of each individual parameter to the model output (Willems, 2012).

The rule-based gate regulation of the adjustable hydraulic structures, that tries to keep the water lev els in a certain range, will not be applied during the sensitivity analysis simulations. Instead, the gate regulation that was simulated by the detailed model, with the default parameter values, will be used as an extra boundary condition. If the rule-based regula tions were applied, the model output variance would be very small, because the change in discharge capac ity is compensated by adjusting the gate levels. There are, however, applications like model predictive con trol (MPC) where gate level regulation and accurate modelling of water levels are of great importance (e.g. Chiang & Willems, 2015).

4 RESULTS AND DISCUSSION

4.1 Performance conceptual model

All conceptual model components are combined into a C code that covers the complete model area.This calcu lation script uses an explicit variable discrete time step calculation scheme and can be run with small calcula tion times in the MATLAB environment (MathWorks, 2012). The model has three boundaries: the upstream Dender discharge originating from theWalloon region, rainfall-runoff discharges distributed over the com plete length of the river and its main tributaries, and the tidally influenced water level in the river Scheldt. To validate the conceptual model, two events (Jan/1995 and Dec/2002–Jan/2003) were simulated and the accuracy of the results evaluated by com parison with those of the detailed MIKE11 model. The parameters of the six considered control structures were chosen arbitrarily, but different from the ones that were used during the calibration. Simulating the two validation events with the conceptual model using a single Intel Core i5-3210M 2.5 GHz PC with 8 GB Ram took on average 11.2 and 22.5 seconds, for respectively Jan/1995 (20 days) and Dec/2002– Jan/2003 (29 days). This is roughly 90 and 70 times faster than the MIKE11 model. The comparison of simulation results of both models is hereafter limited to the water levels in the river Dender, since a detailed comparison of all calculation nodes is practically impossible. For the selected nodes the Nash-Sutcliffe Efficiency (NSE; Nash & Sutcliffe, 1970) and Root Mean Squared Error (RMSE) were calculated, based on time series with an interval of 15 minutes: with $Y_{obs,i}$ and $Y_{pred,i}$ the simulation results of the detailed and conceptual model at time step i, respectively, $Y_{obs,m}$ the mean of detailed model results and n the number of time steps. NSE values lie in a range between 1.0 (perfect fit) and $-\infty$. RMSE values give an idea about the average absolute error between both model results. Both statistical measures are plotted for both validation events in Figure 4. 50% of all calculated NSE values surpass 0.99 and only 10% is lower than 0.92. The water level nodes with the lowest NSE values are located just upstream of the control structures, since the range of water levels is much smaller here than elsewhere along the river. The same phenomenon explains why the NSE values are lower for the Jan/1995 event, which was less severe than the Dec/2002-Jan/2003 event. RMSE values for almost all water level nodes in the river are lower than 5 cm, except for the reaches downstream of the structure at Aalst. Between Aalst and Denderbelle, RMSE values lie around 5 cm, which is still acceptable. Between Denderbelle and Dendermonde, the RMSE augments to about 10 cm. For all water level nodes, the calculated RMSE values are nearly identical for both validation events. This shows that the conceptual model succeeds in providing accurate approximations of the detailed hydrodynamic model results, under different boundary conditions and for different parameter sets. 4.2 Sensitivity analysis To assess the uncertainty in the water levels as a result of the uncertainties in the three considered parameters, the Dec/2002-Jan/2003 event was chosen. The following simulations were performed for this

Figure 4. Statistical comparison simulation results conceptual model vs detailed MIKE11 model.

Figure 5. Water level predictions, approx. 3 km upstream of the control structure in Geraardsbergen, with confidence intervals

(CI): total uncertainty (upper left), critical flow correction factor (upper right), contraction loss coefficient (lower left) and

expan-sion loss coefficient (lower right). The colored lines indicate the default parameter value result.

event: 200 simulations to investigate the individual

influence of each parameter and 600 to investigate

the combined influence. For each Latin Hypercube

Sampling procedure, a sample size of 50 was chosen.

The difference between each analysis is thus only the

number of samples. The sensitivity analysis for one parameter took a bit less than 70 minutes, the com

bined analysis slightly more than 200 minutes. Figure 5

shows the results of this sensitivity analysis for a water

level node in the first reach of the river Dender, which

is usually one of the first to start flooding.

The graphs in Figure 5 indicate that there is a signif

icant influence of the parameters on the water levels

at this point. The 50% and 90% confidence intervals

of the total uncertainty have a mean width of approxi

mately 10.2 cm and 25.5 cm. In fact, the width of these

intervals is not constant: it increases with the discharge

through the structure. This influence is mainly visible when the more downstream located control structure operates in free flow conditions.The peak water levels, on the other hand, are not or only slightly influenced by the considered parameters. During peak flows, the gate level is lowered to its minimum level and the flow through the structure is totally controlled by the downstream water level. This induces important backwater effects to all other water level nodes along the reach. Furthermore, the cross-sectional areas of the structure and the river are close to each other, so that the energy head loss through the structure is low in any case, according to Equation (5). The main parameter contributing to the total uncertainty is the critical flow correction factor, which accounts for about 75% of the total uncertainty. The remaining 25% are chiefly related to the contraction loss coefficient. For this particular location the 90% confidence intervals have a mean width of 25.6 cm

Figure 6. Water level predictions along the river Dender, with RMSE confidence intervals (CI): Total uncertainty, critical

flow correction factor, contraction loss and expansion loss coefficient (from top to bottom). The colored lines indicate the median.

and 7.8 cm for respectively the critical flow correction factor and the contraction head loss coefficient. The influence of the latter is smaller, because the MIKE11 engine already accounts for the fact that the contrac tion head losses are not constant, but a function of the approach velocity.

Figure 6 shows confidence intervals for all water level nodes along the river Dender, between the Wal loon border and the control structure at Denderbelle. The most influenced water levels are found directly upstream of the control structures and the influence diminishes for the more upstream water levels in the reach. The width of all confidence intervals is higher in the more downstream reaches than in the upstream reaches. At the control structure in Aalst, for exam ple, the width of the confidence intervals is twice the width of those at the structure in Geraardsbergen. This is probably due to the combination of uncertainties that arise at the structure and the uncertainties, originating from more upstream, that propagate through the sys tem. Another factor contributing to this observation is the duration of drowned flow conditions, which is shorter for the downstream located structures.

Lowering the critical flow correction factor leads to an increase in the water levels, whereas the oppo site behavior is visible when lowering the contraction loss coefficient. The influence of the expansion loss coefficient is again negligible, when compared to the influence of the other parameters. The maximum width is approximately 2 cm. Between the structures in Idegem and Pollare, the parameter has no influence at all, since the structure in Pollare always operates in critical flow conditions.

Based on the graphs in Figure 5 and Figure 6 it is clear that an accurate assessment of the parameter val ues is of large importance for numerous applications (e.g. MPC). Underor overestimating the discharge capacity can, for example, lead to an incorrect determination of the moment where flooding starts. The methodology presented in this paper can also be used to determine more accurate parameter values, when measurements of gate levels and water levels are available. 5 CONCLUSIONS In this paper a grey-box reservoir-type river model was set up to investigate the sensitivity of river water levels on the parameter values of hydraulic structures. The conceptual model contains the exact same calculation scheme for the hydraulic structures as the detailed model and accounts for the changing curves of the water surface profile. Thanks to the explicit calculation scheme, simulation times are strongly reduced to about 1.3% of the hydrodynamic model simulation time. Furthermore, the time gain is not only achieved by lowering the simulation time, but also by using the MATLAB-environment. The whole sensitivity analysis can be programmed in advance, so that no manual time-consuming interventions in the software package are needed. The methodology was demonstrated for a case study of newly planned control structures in the river Dender in Belgium, making use of a surrogate conceptual model of which the structure was identified and the parameters calibrated to a detailed full hydrodynamic MIKE11 model. The modelling approach is generic, so that any other detailed model, or hydraulic structure calculation scheme can be inserted.

Statistical and visual comparison of the simulation results of both models demonstrate that the conceptual model is capable of reproducing the detailed model results under different conditions, with minor loss of accuracy.

The sensitivity analysis of the three considered parameters shows that the water levels in the river can be highly influenced, especially during low flow con ditions. During extreme peak flows the influence is rather small. The influence of the parameter values is only visible when time series of fixed gate levels are applied to the structures. Water levels are barely influenced when the original gate regulation scheme is applied, since the model will change the gate levels, to meet the regulation conditions.

ACKNOWLEDGEMENTS

The work described in this paper is part of a research study financed by Flanders Hydraulics Research. The authors also thank the Danish Hydraulic Institute (DHI) for providing a license key for the MIKE11 software package.

Chiang, P.-K., & Willems, P. (2015). Combine Evolutionary Optimization with Model Predictive Control in Real time Flood Control of a River System. Water Resources Management, 29(8), 2527–2542. doi:10.1007/s11269 015-0955-5.

DHI (2011). Mike 11, A Modelling System for Rivers and Channels, Reference Manual. DHI Water & Environment, Horsholm, Denmark.

Fenton, J. D. (1992). Reservoir routing. Hydrological Sci

ences Journal, 37(3), 233–246. doi:10.1080/0262666920

9492584.

Helton, J. C., & Davis, F. J. (2003). Latin hypercube sampling

and the propagation of uncertainty in analyses of complex

systems. Reliability Engineering & System Safety, 81(1),

23–69. doi:10.1016/S0951-8320(03)00058-9.

Knight, D., & Shamseldin, A. (2006). River Basin Modelling

for Flood Risk Mitigation. (D. Knight & A. Shamseldin,

Eds.). London, UK: Taylor & Francis Group plc.

MathWorks (2012). MATLAB Help Documentation [Ver

sion r2012a]. The MathWorks Software, Natick, Mas

sachusetts, USA. McKay, M. D., Beckman, R. J., & Conover, W. J. (1979). Comparison of Three Methods for Selecting Values of Input Variables in the Analysis of Output from a Computer Code. Technometrics, 21(2), 239–245. doi:10.1080/00401706.1979.10489755. Meert, P., Willems, P., Nossent, J., Vanderkimpen, P., Pereira, F., Verwaest, T., Mostaert, F. (2016). Development of conceptual models for an integrated catchment management: Subreport 2 – Construction and calibration of conceptual river models. Version 4.0. WL Rapporten, 00_131. Antwerp, Belgium: Flanders Hydraulics Research. Meert, P., Pereira, F. & Willems, P. (2016). Computationally efficient modelling of tidal rivers using conceptual reservoir-type models. Environmental Modelling & Software, 77, 19–31. doi:10.1016/j.envsoft.2015.11.010. Montanari, A. (2007). What do we mean by “uncertainty”? The need for a consistent wording about uncertainty assessment in hydrology. Hydrological Processes, 21, 841–845. doi:10.1002/hyp.6623. Nash, J. E., & Sutcliffe, J. V. (1970). River flow forecasting through conceputal models. Part 1 A discussion of principles. Journal of Hydrology, (10), 282–290. Ngo, L. Le, Madsen, H., Rosbjerg, D., & Pedersen, C. B. (2008).

Implementation and comparison of reservoir operation strategies for the Hoa Binh reservoir, Vietnam using the Mike 11 model. Water Resources Management, 22(4), 457–472. doi:10.1007/s11269-007-9172-1. Syme, W. J. (2001). Modelling of Bends and Hydraulic Structures in a Two-Dimensional Scheme. In Conference on Hydraulics in Civil Engineering. Hobart: The institution of engineers, Australia. Verhoeven, R., Banasiak, R., Okruszko,T., Swiatek, D., Chormanski, J., Nowakowski, P., …Stelmaszcyk, M. (2003). Hydraulic modelling of river flow – data collection and problem solving. Measurements Techniques and Data Assessment in Wetland Hydrology, 1, 9–24. Willems, P. (2012). Model uncertainty analysis by variance decomposition. Physics and Chemistry of the Earth, 42– 44, 21–30. doi:10.1016/j.pce.2011.07.003. Willems, P. (2013). River modelling (in Dutch: Waterloopmodellering). Acco, Leuven & Den Haag, 268 p., ISBN 978-90-334-9296-9, D/2013/0543/132, NUR955. Wolfs, V., Meert, P., & Willems, P. (2015). Modular conceptual modelling approach and software for river hydraulic simulations. Environmental Modelling & Software, 71, 60–77. doi:10.1016/j.envsoft.2015.05.010. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Floods and cultural heritage: Risk assessment and management for the city of Florence, Italy

C. Arrighi & F. Castelli

Department of Civil and Environmental Engineering, University of Florence, Italy

B. Mazzanti

Arno River Basin Authority, Florence, Italy

ABSTRACT: Frequent flood events in the last decades emphasize the unique challenge for urban areas in man

aging natural hazards. Beyond the tangible potential losses to buildings, infrastructures and economic activities,

historic cities have to face the threats on the cultural heritage. International scale reports show that Europe is

on the top of the list for the number of World Heritage Cities (WHC) exposed to high and extreme flood risk.

Increasing risk could lead to a loss of cultural assets that are felt by the population of immense importance

for their contribution to human wellbeing. Nevertheless, much more needs to be done in order to assess flood

risk of cultural heritage. A quantification of the potential losses, direct and indirect, could accelerate the public

debate and stimulate the adoption of protection measures. However, valuing the risk for the cultural heritage is a

complex task since the value of an artwork is hardly monetizable. Here a case study of broad importance is

presented, which is the art city of Florence (Italy), affected by a devastating flood in 1966. This flood event is

considered as exemplary because brought to the world attention the threats posed by natural hazards on cultural

heritage. Currently in Florence 176 buildings are officially classified as being part of the cultural heritage of the

city (e.g. churches, museums, libraries etc.). For an estimated 100-year recurrence interval flood 46 of them may

be affected by the inundation. The number increases to 126 for the 200-year event, similar in magnitude to the

1966 one. Besides the cultural buildings, a huge number of ancient manuscripts, paintings, sculptures and art

objects can be damaged. A preliminary risk assessment is presented for selected damage categories and possible

risk mitigation strategies are discussed.

1 INTRODUCTION

Floods are one of the costliest natural hazards, often

dominating the world annual statistic on catastrophe

losses (Munich Re, 2015). In recent years the atten

tion of the international institutions (i.e. UNESCO) has been focused, beside to the common damage cat egories (i.e. buildings, economic activities), on the potential losses to cultural heritage (European Par liament, 2007; Bigio et al., 2011). Cultural heritage is considered as unique in disaster risk reduction as a symbol of community identity and for its unifying role (UNISDR, 2013). Nevertheless, valuing cultural heritage is a complex task, although it has economic importance because its management state of conserva tion influences human well being and some economic sectors. The knowledge of the cultural buildings and artworks, which are potentially affected in case of flood, is deemed to adopt mitigation strategies specif ically designed for them. At present, no methods suit able for the assessment of flood risk to cultural heritage

in an art city, which has extraordinary concentration of cultural buildings, museums and artworks, have been found in literature. Most of the approaches are theoretical, only a few works provide applications to case studies (Mazzanti, 2003; Lanza, 2003), which are of small extension (i.e. a single monumental complex). This work addresses the issue of estimating flood risk to cultural asset in an exemplary art city, which is the city of Florence (Italy). Florence was severely affected by a flood in November 1966, which caused priceless damages to ancient manuscripts, paintings and frescoes beside the monetary damages to structures, infrastructures and economic activities. Thus, it became an exemplary case for flood risk managers in Italy. The hydrologic-hydraulic model and the damage assessment method for tangible losses has been described in Arrighi et al. (2013). Here several damage categories for cultural buildings and artworks are selected and the risk to cultural heritage is estimated in absence of risk mitigation strategies (Arrighi et al., 2016). Finally the cultural heritage most at risk is mapped in terms of annual expected losses and possible management strategies for the UNESCO site are discussed.

Figure 1. Damages of the 1966 flood on the wooden Cimabue's crucifix (XIII century), which became the symbol of the artworks losses.

2 RISK ASSESSMENT FOR CULTURAL HERITAGE

2.1 Hydraulic model

As the first step, the inundated areas and flood depths are identified and mapped for different flood sce narios using a coupled 1D/quasi-2D hydraulic model described in Arrighi et al. (2013). The computation of the flood propagation and corresponding water profile along the river is performed through a standard solver of the 1-D general equation of unsteady flow (i.e. con tinuity and momentum conservation equations). The quasi-2-D hydraulic model for the floodplain consist of several storage areas (cells) whose effective geom etry is estimated from a meter-scale DSM. Buildings are, by default, considered as waterproof blocks. The continuity and momentum conservation equations are solved through an implicit 1D finite difference scheme for the river. The outflow from the river banks is mod elled through a set of lateral weirs connecting river and floodplain. When the inundation starts, the quasi

2D module, governed by continuity and stage-storage relations, calculates the water levels from the vol ume stored in the cell. Flow between adjacent cells is described by a weir equation accounting for backwater effects.

2.2 Risk to cultural heritage

In the view of assessing the risk to cultural heritage in a whole city, which is a UNESCO site, the method here Figure 2. Sketch of the method for flood risk assessment to cultural heritage. proposed glances at the common definition of flood risk but with some necessary adjustments. A distinction should be made between movable and immovable cultural asset. In fact, an historic building may contain or not artworks and their number at risk is not necessarily proportional to its areal extension or geometric characteristics. This conceptual difference is functional to the implementation of risk management plans and design of mitigation measures. Thus, for the assessment of the potential losses, the category of the cultural heritage is divided into buildings (e.g. the containers) and artworks (e.g. contents) (Tab. 1). The risk assessment method is represented in the diagram of Figure 2. Hazard and exposure are evaluated through the intersection between the GIS layers of the cultural buildings (Fig. 2) and inundated areas for different scenarios. The heritage building/artwork is considered as affected if lies inside an inundated area. The number and vulnerability class of exposed artworks in each cultural building is identified through a spatial join between the buildings shapefile and artworks recognition sheets (Autorità di Bacino del Fiume Arno, 2013).

Table 1. Damage categories for cultural buildings and art works (left column) and assigned vulnerability level (right column).

Buildings	Vulnerability
Churches and religious buildings	high
Libraries, archives	low

Museums medium

Noble palaces, theatres medium

Artworks Vulnerability v_{art}

Paintings .60

Books, art prints 0.60

Sculptures 0.15

Goldsmith's art 0.10

2.3 Risk assessment method for artworks

Avoiding the monetary assessment of the historic buildings and artworks, a simplified risk quantification $R_{cult,art}$ should return the number of artworks which are lost or very seriously damaged in any given year. This results from the actualization of the number of lost objects weighted by the return period T_R of the flood event

where $L_{cult,art}$ is the number of exposed artworks lost for a given scenario,

where i is the artworks damage category (Tab. 1), k is the number of categories, $v_{art,i}$ is the degree of damage (i.e. the vulnerability) and $n_{art,i}$ is the number of artworks inside the cultural building.

2.4 Risk assessment method for cultural buildings

For the buildings, a qualitative classification based on three vulnerability classes is proposed. Risk for cultural buildings $R_{cult,b}$ is classified according to a risk matrix which accounts for hazard level H and potential damages $D_{cult,b}$ (Tab. 2). The risk matrix has been derived assigning to hazard and damages an integer score from 1 (low) up to 3 (high). Poten tial damages depend on vulnerability levels, which are assessed referring to categories sharing homoge neous characteristics (i.e. religious buildings) in the study area.

3 RESULTS

3.1 Inundation scenarios and exposure

The Arno river catchment is located in central Italy is characterized by an area of 9116 km^2 and the river length is 241 km. The average elevation of the catchment is 353 m.a.s.l., 166 municipalities lie in its territory for a total number of 2.2 millions inahbitants. Florence is the main urban agglomeration and is geographically placed at the middle of theArno length. The hydrologic regime shows a great difference between minimum and maximum mean-daily discharges. Annual peak discharges for the most downstream gauging station (S. Giovanni alla Vena) range from 321 to 2290 m^3/s (recorded on November 4, 1966). The maximum flow discharge, reached on November 4, 1966, just upstream the Florence urban reach of the Arno river has been estimated about 4100 m^3/s. Hydrologic-hydraulic data and high resolution GIS data for the model set-up are provided by the Arno River Basin Authority (Autorità di Bacino del fiume Arno, 1999) and by the Municipality of Florence. The estimated peak discharge for the 100 and 200 years are 3400 m^3/s and 3900 m^3/s respectively. For the 30 years scenario the inundation extent is limited and no cultural heritage is involved since a sub-urban district is affected. In the 100 years scenario the flooded area is about 3 km^2 (Fig. 2, panel a) with an average flood depth of 2 m. In the 200 years scenario, flooded area and depth

Table 2. Risk matrix for cultural buildings.

	$D_{cult,b}$ Low	$D_{cult,b}$ Medium	$D_{cult,b}$ High
H (T_R > 200 years)	Low	Low	Medium
H (30 < T_R ≤ 200 years)	Low	Medium	High
H (T_R ≤ 30 years)	Medium	High	High

increase to 11 km 2 (Fig. 2, panel a) and 2.5 m respectively (Arrighi et al., 2013). The estimated pattern of the 200 years flood in Florence shows strong similarities with the historic flood of 1966 as described in Arrighi et al. (2013). In the municipality of Florence 176 building are officially classified as being part of the cultural heritage of the city (e.g. churches, museums, libraries etc.). For the 100 years flood scenario 46 of them are affected by the inundation, for the 200 year event the number increases to 126 (about 70% of total) (Fig. 1, panel a). 3.2 Cultural buildings The vulnerability of cultural buildings has been qualitatively assessed given the substantial homogeneity inside the same building category in the case study area. The highest vulnerability has been assigned to churches and religious complexes, a medium and low vulnerability has been assigned to museums, noble palaces, theatres and libraries respectively (Tab. 1). Among the cultural heritage of the municipality, counting 176 cultural buildings, 46 are classified at high risk and 60 at medium risk.

Figure 3. Exposed cultural heritage superimposed over the

flooded area for the 100 years scenario and 200 years scenario

(panel a) and detail of flood risk to artworks as relative annual

loss with respect to the total number of exposed objects for

the city centre of Florence (panel b).

3.3 Artworks

The available reports on the number and quality of

restored artworks affected in 1966 have been studied

to assess the percentage of losses for different artworks

classes and estimate L cult,art (Eq. 2) for each scenario

(i.e. 30, 100, 200, 500 years of recurrence interval)

(Autorità di Bacino del Fiume Arno, 2013).

The main assumptions underlying the risk assess

ment to cultural heritage are have been discussed by

Arrighi et al., 2016.

According to the reports on the damages of the 1966

flood (Batini, 1967; Sebregondi, 2009; Corradetti,

2014, Giusti, 2015) the most vulnerable artworks are

in descending order paintings and books (v art = 60%),

sculptures (v art = 15%), goldsmith's art (v art = 10%)

(Tab. 1). In the map in Figure 3 panel b, a color scale from green to red represents the annual expected percentage of artworks lost (over the total exposed artworks in each building). The annual percent losses range from 0.01% up to 1.3%. The 0% damage corresponds to the cultural buildings, which do not contain artworks in the floors potentially inundable, i.e. the exposure is zero. The highest number of absolute annual losses are expected for S. Croce church and cloister (in red in the bottom right side of Fig. 3, panel b) and for the Uffizi Gallery, which hosts a deposit at the ground level. The cultural buildings containing the top three number of relative annual losses are shown at the bottom of Figure 3 (panel b), they are a church (1), a library (2) (visible in the panel a) and the Uffizi Gallery (3). Although it may appear surprising, there are less artworks at risk close to the river. In fact, after the 1966 flood some collections were moved to safer exhibition areas, thus implementing non-structural mitigation measures. For several cases, such as the S. Croce church, the artworks are still considered site-specific, and the debate on protection measures is ongoing. Cultural heritage resulting at high risk (Arrighi et al., 2016) both as a building both for the contained artworks suggest that a thorough attention should be paid to its protection. On average, the highest global risk is found for religious buildings, which usually store precious art pieces and are characterized by vulnerable structural details. 3.4 Mitigation strategies The survey on the cultural heritage in Florence (Autorità di Bacino del fiume Arno, 2013) also includes the number of specialized teams (three trained personnel units each) required to protect the movable artworks in case of flood alarm. In fact, among the debated emergency measures that have been taken into consideration, the most effective for movable artworks is the quick relocation to upper floors, i.e. to a safer elevation, but this is not always possible. The relative amount of artworks that in each building can be moved to a safe elevation in shown in the map of Figure 4, which shows the ratio between movable and total number of exposed artworks. The values of this ratio can be equal to one (especially for libraries), however this may require a large number of teams (Castelli, 2014). For a flood with an estimated recurrence interval of 100 years the required number of teams is about 80, which significantly increases to about 500 in case of an event of magnitude similar to the 1966 flood (i.e. approximately 200 years recurrence interval).This kind of activity obviously requires periodical training activities and warning simulation besides a robust organization and efficient flood prediction. For the heritage buildings in the map of Figure 4, where the ratio between movable and total number of artworks is low, different management strategies should be adopted, such as the installation of sealing

Figure 4. Ratio between movable artworks and total number

of exposed artworks for a detail of the historical district, only

the buildings containing artworks are represented.

waterproof coats. This kind of measure is suitable

for sculptures and statues (Acidini, 2014) when their

dimensions and weight are too large to ensure a safe

displacement.

The protection of cultural buildings can be achieved

waterproofing the basement or underground floors

through specific retrofitting strategies, such as the

installation of backwater valves to avoid inflows from

the sewer system.

4 CONCLUSIONS

The result of the adopted approach suggests that, in

case of absence of precautionary actions, each year

about 0.11% of exposed artworks may be lost on average in the city of Florence. An event of a magnitude similar to 1966 flood could cause a 40% of loss among the exposed artworks. Similar to that for economic activities, many artworks were moved to safer positions (although some of them were definitively lost). Therefore, a direct comparison between estimated and experienced cultural heritage losses is not possible. The risk assessment is highly conservative and assumes that no emergency plan exists to protect the cultural heritage of the city. Moreover, the development of innovative restoration methods, which could save more artworks compared to 1966, is not considered.

This work wants to bring to the fore that different damage categories and damage assessments provide vital information to support flood risk management.

In the case of a whole city, the economic evaluation of flood risk is crucial for the design of catchment scale mitigation strategies (Arrighi et al., 2016). Cultural

Giusti, A. 2015. Il paradiso ritrovato, il restauro della Porta del Ghiberti, ISBN 9788874612475 Mandragora Editore, Firenze.

Lanza, S.G., 2003. Flood hazard threat on cultural heritage in the town of Genova (Italy). Journal of Cultural Heritage, 4, 159-167.

Mazzanti, M. 2003. Valuing cultural heritage in a multi

attribute framework, microeconomic perspectives and

policy implications. Journal of Socio-Economics, 32,

549–569. Munich Re 2015. Loss events worldwide 1980 – 2014. 10 costliest events ordered by overall losses. Sebregondi, L., 2009. Il Museo dell'Opera di Santa Croce a Firenze, ISBN 9788874611379 Mandragora Editore, Firenze. UNISDR 2013 Heritage and resilience-Issues and opportunities for reducing disaster risks. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Assessing the impact of climate change on extreme flows across

Great Britain

L. Collet & L. Beevers

Heriot-Watt University, Edinburgh, UK

C. Prudhomme

CEH, Wallingford, UK

ABSTRACT: Climate projections are predicted to result in the increase of UK properties at risk from flooding.

It thus becomes urgent to assess the possible impact of these changes on extreme high flows in particular, and

evaluate the uncertainties related to these projections. This paper aims to assess the changes in extreme runoff

for the 100-year return level across Great Britain. It is based on the Future Flow database and analyses daily

runoff over 1961–2098. The Generalized Extreme Value (GEV) distribution function is automatically fitted over

the baseline and the 2080s for 281 gauging stations. The analysis evaluates the uncertainty related to the GEV,

and to the climate model parameterization. Then it assesses return levels with combined uncertainties. Results

show that the west coast of the whole country presents the lowest climate model and probabilistic distribu

tion uncertainties. The climate model parameterization provides greater uncertainty than the GEV distribution

function.

1 INTRODUCTION

1.1 Flood risk in the UK and climate change

Floods are the most common and widely distributed natural risk to life and property worldwide. The UN estimate that 1B people live in areas of potential flood risk (Jonkman & Vrijling, 2008), and damage caused by floods has been significant in recent decades. Worldwide, floods resulted in over $16Bn in dam ages and 1500 fatalities in 2014 alone (Guha-Sapir, et al. 2015). In the UK over the past five years there have been significant challenges to water management posed by flooding. Since 2000, flooding has caused over £5Bn worth of damage (ABI, 2013), of which £3Bn was caused by the 2007 floods, and over £1Bn from the 2013/14 winter storms. Climate projections are predicted to result in the increase of properties at risk from flooding and thus understanding the uncer tainty associated with future climates to flood hazard is of vital economic significance to the UK. Looking forward the UK government has estimated that each year damages of £1.1Bn will arise from flooding, and that maintaining the current levels of flood defense will require up to £1Bn per year by 2035 (UK Parliament, 2013). Understanding the uncertainty associated with climate change to future extreme events is therefore of vital economic significance to the UK government. 1.2 A need to evaluate uncertainties related to climate change One of the biggest challenges to successful flood risk management is understanding the extent to which climate change will influence exposure to flood events. Flood risk management has historically approached flood hazard prediction in a deterministic manner (e.g. Di Baldassarre et al., 2010; Balica et al., 2013). However, this approach omits the uncertainty due to the dynamic, stochastic and uncertain nature of the climate, hydrological and river processes, the limitations in quantifying model performance and the assumptions embedded within the analysis. Recent research (Beven and Hall, 2014) has looked to account for uncertainty in modeling approaches; however this work has not explicitly included climate change projections as a source of uncertainty. Accounting for the uncertainties in model prediction, including climate change, necessitates incorporating ensemble model inputs which recognize the uncertainties in the prediction process (von Christierson et al., 2013). It is essential therefore that flood risk projections adopt such an approach to deal with the propagation of these uncertainties through the modeling process. Climate model outputs present uncertainties related to the model structure, its parameterization, the chosen emission scenario, and, for regional and local impact studies, the downscaling method used (Prudhomme et al., 2003; Wilby, 2010). There is a clear need to understand climate model uncertainty and quantify the impact this may have on flood risk. As a first step, the uncertainty in extreme event prediction, associated with climate projection, requires to be quantified.

1.3 Aims of this paper

This work is based on the UK national scale Future

Flows database, which presents a unique opportunity to investigate, in a consistent manner, the influence of climate projection uncertainty on flood hazard esti mation across Great Britain. Extreme value theory is commonly used to assess extreme return period flows, generally with the GEV distribution. Consequently this paper aims to investigate climate model and Extreme Value distribution uncertainties to extreme flow prediction across the UK for the 100-year return period runoff.

2 MATERIAL & METHOD

2.1 The Future Flow database

The UKCP09 (Murphy et al., 2009) provides various downscaled scenarios across the UK, and based on this data the Centre for Ecology and Hydrology (CEH) led a nation-wide study to assess river flow and ground water scenarios for 11 climate-change ensembles over 1961–2098 (Future Flow project, Prudhomme et al., 2013). These ensembles reflect the uncertainties related to the HadRM3-PPE-UK model parameteri zation, as derived from the UKCP09 scenarios under the A1B emission scenario. The datasets provide a set of climate projections across Great Britain at suitable spatial and temporal resolution for hydrological stud ies, and present a unique opportunity to investigate the potential uncertainty associated with climate mod

els to extreme flows across the UK. This information

can then be cascaded through numerical models to

understand changing flood hazard. Daily river flows

are provided for 281 river catchments. This study is

based on the Future Flow Hydrology database for

daily river flow and is developed on two characteristic

time periods: the baseline (from January 1961 to

December 1990) and the 2080s (from January 2069

to December 2098).

2.2 The GEV distribution function

To analyze the hydrological dataset this study is

based on non-stationary extreme value analysis (Coles,

2001), specifically the Generalized Extreme Value

(GEV) distribution for block maxima (Eq. 1) to cre

ate distributions of predicted return levels for different

time periods from the ensemble data. The extreme

value theory considers (X_1, X_2, . . ., X_n) a sequence

of independent random variables and $M_n = \max\{X_1$,

X_2, . . ., $X_n\}$ a sequence of block maxima (Coles, 2001). If there are $a_n > 0$ and b_n, such that the probability: Then G(z) belongs to the Generalized Extreme Value family distribution: where μ= location parameter; $\sigma > 0$ = scale parameter; ξ = shape parameter. Although it would be ideal to use the GEV on a 100-year time period, there is a risk that such a time period would be non-stationary and thus induce a bias. In the context of climate change, 30-year time periods are defined as stationary and non-stationarity induced by climate change is investigated by comparing results on past and future periods and assessing changes from one period to another. In this study, the distribution function is fitted to the daily flow database across Great Britain for the 11-member ensemble over the baseline and the 2080s (2069–2098) time periods. Once fitted, it is used to assess flow for the 100-year return period. 2.3 Assessing uncertainties related to the GEV distribution function and climate change model variability To understand the spatial variations of extreme flows for the 100-year return period, data series of daily runoff (in mm, river flow divided by the catchment area and multiplied by 24 * 3600 seconds) are fitted to the GEV distribution function. The analysis is carried out in three main steps: (i) The first step investigates the uncertainty related to the GEV function parameterization. For only one climate ensemble, the 100-year return level and its 95% confidence interval are assessed. To ensure a normative comparison across Great Britain, and thus to remove the bias related to larger or smaller rivers, a relative coefficient of uncertainty is computed (Eq. 2): where E = runoff estimate; CI up = upper 95% confidence interval; CI low = the lower 95% confidence interval. Similarly to the relative standard deviation (Eq. 3), this coefficient is dimensionless and allows the comparison of data with different means and confidence interval ranges. The result map gathers the information related to the estimated runoff and the coefficient of variability for each station.

(ii) The second step investigates the uncertainty related to the climate model parameterization. Return levels are computed for the 11 climate-change ensembles, resulting in a Cumulative Distribution Function (CDF) for the baseline and the 2080s. The result map for Great Britain presents the median estimate and the relative standard deviation for each station (Eq. 3).

where σ = standard deviation; μ= mean of the distribution.

The relative standard deviation is a standardized measure of dispersion of a probability distribution.

It is a dimensionless number, expressed in percent age, which allows the comparison between data with different means and dispersions.

(iii) The third step consists in combining both sources of uncertainty and assessing return level esti mates with the GEV function for the 11 climate change ensembles. Combined uncertainties are presented through a CDF of the estimates and their associated 95% confidence intervals. Results are pre sented for seven contrasted gauging stations.

3 RESULTS

3.1 Uncertainty to the GEV function parameterization

Figure 1 shows the estimated past and future 100-year return period runoff for one climate-change ensemble and the associated 95% confidence interval (step 1) calculated across Great Britain with the GEV dis tribution (Fig. 1a for baseline and Fig. 1b for the 2080s). The color range represents the variation in estimate. The circle scale represents the variation in GEV distribution uncertainty (relative CI variance, Eq. 2).

Higher values of estimates are located in the west of Great Britain, especially in Scotland (e.g. up to 101 mm for river Falloch at Glen Falloch), while the eastern side presents lower values, especially in south east of England (e.g. 1.7 mm for river Mimram at Panshanger Park).The 100-year return period estimate tends to increase from the baseline to the 2080s mostly in the west: in west Scotland (e.g. 78 to 111 mm for river Carron at New Kelso), Wales (e.g. 54 to 94 mm for river Ystwyth at Cwmystwyth), and southwest of England (e.g. 46 to 72 mm for river Barle at Brush ford). It decreases particularly in northeast of Scotland (e.g. 54 to 18 mm for river Nairn at Firhall) and north east of England (e.g. 27 to 17 mm for river Ouse at Skelton).

Similarly, the GEV distribution computes the low est CI relative variances on the west coast (down to 3% in west Scotland for river Falloch) and the highest in southeast England (up to 43% for river Gipping). The

uncertainty related to the GEV distribution increases from the baseline to the 2080s mostly in Scotland (e.g. 32 to 42% for river Leet Water at Coldstream), central England (e.g.) and the west coast of Wales. It decreases particularly on the northwest coast of England (Carlisle and the Eden river), and the southeast of England (chalk catchments). Note that these results are shown only for one climate-change ensemble and do not represent the whole climatic variability at this stage. In summary, when one climate-change ensemble only is considered, the GEV distribution computes: • Higher 100-year return period estimates with lower GEV uncertainty in the west of Great Britain; • Greater GEV uncertainty with lower estimates in south and southeast of England; • Increased runoff between the baseline and the 2080s. 3.2 Uncertainty to the climate model parameterization Figure 2 shows the estimated past and future 100-year return period runoff for the 11 climate-change ensembles and the associated uncertainty (step 2) calculated across Great Britain with the GEV distribution (Fig. 2a for baseline and Fig. 2b for the 2080s).The color range represents the variation in the mean estimate. The circle scale represents the variation in climate model uncertainty (relative standard deviation, Eq. 3). At the national scale, there is a clear delineation between the highest runoff values on the west coast (e.g. 97 mm for river Falloch at Glen Falloch) and the lowest on the east side of Great Britain (e.g. 5.35 mm for Stanford Water at Buckenham Tofts). From the baseline to the 2080s, the situation remains quite similar over the whole country, with an increase in mean estimates for various stations across the country (e.g. from 88 to 96 mm for river Carron at New Kelso). The climate model uncertainty suggested by the GEV is the lowest along the west coast of Great Britain (e.g. 6% for river Falloch at Glen Falloch), and highest on the east coast, especially in southeast England (e.g. 75% for river Belchamp Brook at Bardfield Bridge). There is a net increase in uncertainty in northeast, southeast and south of England (e.g. from 16 to 97% for river Kym at Meagre Farm), and a decrease for some stations in north Scotland (from 27 to 12% for river Meig at Glenmeanie), and west (from 33 to 18% for river Dane at Rudheath, Dee catchment) and south (from 69 to 23% for river Boyd at Bitton, Avon catchment) England. In summary, when considering the 11 climatechange ensembles, the GEV computes: • Higher 100-year return period estimates with lower climate model uncertainty on the west coast; • Higher climate model uncertainty with lower 100-year return period estimates particularly in southeast of England;

Figure 1. Uncertainty to the GEV distribution across Great

Britain: 100-year return period runoff for one climate-change

scenario and relative coefficient of uncertainty associated to

the 95% confidence interval estimation on: (a) the baseline;

(b) the 2080s. Table 1. For each selected gauging station, on baseline and the 2080s: uncertainty related to climate model (CM U) and uncertainty to the GEV distribution (GEV U). CM U (mm) GEV U (mm) Station baseline 2080s baseline 2080s Don River 10.81 12.08 5.06 10.24 Ribble River 10.09 27.19 23.33 28.94 Thames River 3.76 4.71 1.69 4.6 • Increased climate model uncertainty between the baseline and the 2080s especially in southeast of England. 3.3 Combined uncertainties on contrasted catchments Figure 3 shows the estimated past and future 100return period runoff for the 11 climate-change ensembles calculated with the GEV distribution and the associated 95% confidence intervals (step 3) for the baseline and the 2080s. Three gauging stations representative of their region were chosen across Great Britain: the Don River in Scotland (Fig. 3a); the Ribble River in north England (Fig. 3b); and the Thames River in southeast England (Fig. 3c). For each station results calculated are in black and grey for the baseline and in blue for the 2080s. For all the stations, there is an increase in runoff estimate from baseline to the 2080s. The uncertainty related to the GEV distribution is always higher for larger estimate values, and globally highest for the station with the highest climate model uncertainty (Fig 3b). The highest estimates are found on the western station (the Ribble Fig. 3b), while the southern station presents the lowest estimates (the Thames Fig. 3c). The uncertainty related to climate model increases from baseline to the 2080s especially for the Ribble (Fig. 3b). Similarly, the uncertainty related to the GEV distribution increases notably from baseline to the 2080s for this station. Table 1 presents the uncertainty related to the climate model (CM U) and the uncertainty related to the GEV distribution (GEV U) for each station in Figure 3. To quantify these uncertainties, the difference between (i) the maximum and minimum estimates and (ii) the minimum lower and maximum upper 95% confidence interval limit minus the CM U were computed respectively. The uncertainty related to climate model increases from baseline to the 2080s, for the three stations. The uncertainty related to the EV distributions also increases from baseline to the 2080s for all the stations. The uncertainty related to the climate model is of similar magnitude to the GEV distribution uncertainty. The Don and the Thames Rivers present a higher CM U than to the probabilistic distribution, while for the Ribble River this is reversed.

Figure 2. Uncertainty to the climate model parameteriza

tion across Great Britain: mean 100-year return period runoff

with the GEV distribution and relative standard deviation

associated to the 11 climate-change ensembles: (a) baseline;

(b) the 2080s. Figure 3. Combined uncertainties related to GEV distribution and climate model parameterization on three contrasted catchments: cumulated distribution function of the 11 100-year return period estimates and associated 95% confidence intervals over the baseline and the 2080s for (a) the Don River at Parkhill; (b) the Ribble River at New Jumbles Rock; (c) the Thames River at Eynsham.

4 DISCUSSION

4.1 Main results

Across Great Britain (Fig 1, 2 and 3) the GEV distri bution generally suggests spatially contrasted runoff estimates with the highest values on the west coast, and the lowest associated climate model and proba bilistic distribution uncertainties. Similarly, the lowest estimates are found in south and southeast England, with the highest uncertainties of the country. From the baseline to the 2080s horizon, the GEV distribution shows increasing distribution function uncertainty and climate model uncertainty.

The spatial contrast between the west coast and the southeast of the UK is correlated to the climatic and geological conditions. There is a strong northwest-southeast rainfall gradient (see e.g. Stewart et al., 2014), with the highest 1:100 year return levels in northwest of Scotland, England and Wales (up to 400 mm), and the lowest in south and south east of England (down to 60 mm). Moreover, the main aquifers are located in southeast of England, with the chalk and Jurassic limestone reservoirs. This context induces a significant variability in catchment con ditions and hydrology: on the west coast catchment responses are more constrained by the rainfall variability while in southeast the groundwater contribution is more significant.

4.2 Dissociated uncertainties

Figure 4 investigates the proportion of uncertainty related to the parameterization of the climate model (CM) and the GEV distribution function for all the gauging stations across the UK studied in this work. Results for the baseline are shown on the top row and on the bottom row for the 2080 horizon.

The proportion of uncertainties induced by the CM (in black) and the GEV distribution (in grey) are similar between the baseline and the 2080s, and varies greatly between the gauging stations (e.g. from 99.96/0.04% to 24.30/75.70%; proportion of CM/GEV uncertainty). Generally, the CM parameterization provides the greater uncertainty (58% and 59% on the baseline and the 2080s respectively). The GEV distribution seems thus globally robustly defined for the 100-year return period estimation and quite sensitive to the input data it is fitted on.

As the proportion of GEV distribution and climate model uncertainty is variable across the catchments, these uncertainties should systematically be combined and accounted for when assessing high return-period flows.

4.3 Limits

Some limits of this study are related to the use of

the Future Flow database. As reported by Prudhomme

et al. (2013), three models are used to simulate river

flow (CERF, Griffiths et al., 2006; PDM, Moore,

2007; and CLASSIC, Crooks & Naden, 2007) with the Figure 4. Relative Climate Model (CM) and Generalized Extreme Value Distribution (GEV) uncertainties for each gauging station across the UK over the baseline (top) and the 2080s (bottom). White figures: mean total proportion across the UK stations of CM (top) and GEV (bottom) uncertainty. emphasis of calibration on different parts of the flow regime. For CERF the emphasis is on water resources as represented by the water balance and low flows, while for PDM and CLASSIC the emphasis is on the upper part of the flow regime and peak flows. For the gauging stations calibrated with the CERF model, the estimated 100-year return period might thus be under-estimated (for 221 gauging stations). Moreover, this study investigates the uncertainty related to one climate model only (HadRM3). Using a wide range of General Circulation Models (GCMs) outputs is generally recommended to estimate a set of possible futures in impact studies, as well as the probability of occurrence of these future scenarios (Wilby, 2010). With such a probabilistic approach, uncertainty of climate models outputs can be characterized and sensitivity and performance of impact models evaluated based on these uncertainties. However as reported by Ludwig et al. (2014) these uncertainties can range widely, related to divergent GCMs outputs with large biases in simulation, especially precipitation, resulting in little concluding information in impact studies. 5 CONCLUSION This study has used the Future Flow dataset to analyse the potential change to the 100-year return period flow in the 2080s over the baseline. This analysis was carried across Great Britain on 281 catchments using the GEV distribution function.

Results showed higher 100-year runoff estimates

on the west coast and lower in southeast of England.

The uncertainties related to the GEV distribution and the climate model parameterization are both lower on the west coast and higher mainly in southeast of Eng land. From the baseline to the 2080s, the 100-year return period estimates generally increase, as well as the climate model uncertainty.

Analyzing the proportion of uncertainty from the GEV and climate model sources showed the impor tance of taking both of them into account. Although the climate model globally induces higher uncertainty than the GEV distribution, this trend is uneven across British catchments and the combination of both uncer tainties should be accounted for when assessing runoff return levels.

Finally, other distribution functions from the extreme value theory could also be tested, such as the Generalized Pareto Distribution (Coles, 2001). The 100-year return period estimate and related uncertainty computed with this distribution could be assessed and compared to the GEV results.

Association of British Insurers (ABI), 2013. www.abi.org Accessed 26th October 2015.

Balica, S. Beevers, L., Popescu, I., & Wright, N. 2013. Para metric and physically based modelling techniques for flood risk and vulnerability assessment: a comparison; Journal of Environmental Modelling & Software, 41(3):

81–92.

Beven, K., Hall, J (eds). 2014. Applied uncertainty analysis for flood risk management. Imperial College press ISBN 978-1-84816-27-9.

Coles, S. 2001. An Introduction to Statistical Modelling of Extreme Values. Springer-Verlag, London.

Crooks, S. M. & Naden, P. S. 2007. CLASSIC: a semi distributed rainfall-runoff modelling system, Hydrol. Earth Syst. Sci., 11, 516–531.

Di Baldassarre, G. Schumann, G. Bates, P. Freer J. & Beven K. 2010. Flood-plain mapping: a critical discussion of deterministic and probabilistic approaches. Hydrological Sciences Journal, 55(3): 364–376.

Guha-Sapir, D., Below, R., & Hoyois, P. (2015). Natural dis asters reported. EM-DAT: international disaster database, Université Catholique de Louvain, Brussels, Belgium.

Griffiths, J., Young, A. R., and Keller, V. 2006. Continuous Estimation of River Flows (CERF) – Technical Report: Task 1.3: Model scheme for representing rainfall interception and soil moisture, CEH, Wallingford, 45 pp.

Jonkman, S. & Vrijling, J. 2008. Loss of life due to floods. Journal of Flood Risk Management, 1:43–56.

Ludwig, F., van Slobbe, E., Cofino, W. 2014. Climate change adaptation and Integrated Water Resource Management in the water sector. Journal of Hydrology. 518, 235–242.

Moore, R. J. 2007. The PDM rainfall-runoff model, Hydrol. Earth Syst. Sci., 11, 483–499.

Murphy, J.M., Sexton, D.M.H., Jenkins, G.J., Boorman, P.M., Booth, B.B.B., Brown, C.C., Clark, R.T., Collins, M., Harris, G.R., Kendon, E.J., Betts, R.A., Brown, S.J., Howard, T. P., Humphrey, K.A., McCarthy, M. P., McDonald, R. E., Stephens, A., Wallace, C., Warren, R., Wilby, R., Wood, R. A. 2009. UK Climate Projections Science Report: Climate change projections. Met Office Hadley Centre, Exeter.

Prudhomme, C., Jakoba, D. & Svensson, C.

2003. Uncertainty and climate change impact on the flood regime of small UK catchments. Journal of Hydrology, 277: 1–23. Prudhomme, C., Haxton, T., Crooks, S., Jackson C., Barkwith, A., Williamson, J., Kelvin, J., Mackay, J., Wang, L., Young, A. & Watts, G. 2013. Future Flows Hydrology: an ensemble of daily river flow and monthly groundwater levels for use for climate change impact assessment across Great Britain. Earth Syst. Sci. Data, 5: 101–107. Stewart L, Vesuviano G, Morris D, Prosdocimi I. 2014. The new FEH rainfall depth-duration-frequency model: results, comparisons and implications. Presentation at the 12th British Hydrological Society National Symposium. Birmingham, UK, 2-4 Sept 2014. Available at: http://nora.nerc.ac.uk/510563/ UK Parliament. 2013. Current parliamentary material available on flooding http://www.parliament.uk/topics/ Flooding.htm accessed 16th August 2013 von Christierson, B, Wade, S, Counsell, C, Arnell, N, Charlton, M, Prudhomme, C, Hannaford, J, Lawson, R, Tattersall, C, Fenn, C & Ball, A. 2013. Climate change approaches in water resources planning – overview of new methods. Environment Agency, Report – SC090017/R3, ISBN: 978-1-84911-291-8. Wilby, R. L. (2010). Evaluating climate model outputs for hydrological applications. Hydrological Sciences Journal, 55(7), 1090–1093. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Continuous hydrologic modeling of coastal plain watershed using

HEC-HMS

M. Pourreza-Bilondi

Department of Water Engineering, University of Birjand, Birjand, Iran

S. Zahra Samadi

Department of Civil and Environmental Engineering, University of South Carolina, Columbia,

South Carolina, USA

ABSTRACT: This paper examines the effects of hydrologic model parameterization uncertainty, where multi

ple unique hydrologic model parameters sets can result in adequate calibration metrics, on hydrologic modeling of the Waccamaw watershed (3116 km 2). HEC-HMS is linked to the generalized likelihood uncertainty estima

tion (GLUE) and is calibrated for daily streamflow at two USGS gauging stations during 2003–2005 period.

A standard conceptual HEC-HMS's soil moisture accounting (SMA) approach is applied in this study that repre

sents a sub-basin with well-linked storage layers/buckets accounting for canopy interception, surface depression

storage, infiltration, evapotranspiration, as well as soil water and groundwater percolation. Analysis demon

strated that 95 prediction uncertainty bracketed most high and medium flows while slightly overestimated the

magnitude and direction of low flow events. Since a large portion of the runoff-producing in the coastal plain

watershed is groundwater dependent, hydrologic model parameterization may lead to statistically significant

differences in streamflow prediction during different timescales.

Keywords: Uncertainty Analysis; GLUE; HEC-HMS; Coastal Plain Watershed.

1 INTRODUCTION

Uncertainty assessment helps enhance the reliabil

ity and credibility of the hydrological outputs and

provides reliable information for effective decision

making and water resources management. Hydro

logic prediction is the only way to estimate watershed

parameters, since most of watershed parameters can

not be inferred through direct observation in the field,

but can only be meaningfully derived by calibra

tion against an input-output record of the catchment response (e.g. Vrugt et al., 2008).

The goal of model calibration in hydrology has been focused on finding values for the immeasur able parameters that result in the best fit to the observed data, especially streamflow parameters. In its most elementary method, the calibration is performed by manually adjusting the parameters while visually inspecting the agreement between observations and model predictions (Janssen and Heuberger, 1995). In this approach, the “closeness” of the model with the measurements is typically evaluated in terms of sev eral subjective visual measures, and a semi-intuitive trial-and-error process is used to perform the param eter adjustments (Boyle et al., 2000). But subjectivity and time-consuming nature of this method causes a great deal of research into the development of automatic (computer based) methods for calibration of hydrologic model.Automatic calibration encompasses a wide range of different calibration methodologies, from global optimization (Duan et al., 1994; Gan and Biftu, 1996;Yapo et al., 1996) to Monte Carlo method (Beven and Binley, 1992; Freer et al., 1996; Uhlenbrook et al., 1999), with additional consideration have been given to multi-objective methods (Yapo et al., 1998; Vrugt et al., 2003b). The goal of each of the aforementioned automatic calibration approaches is to reduce the subjectivity common in manual calibration by implementing mathematical functions to evaluate goodness-of-fit (Vrugt et al., 2003a), while at the same time taking advantage of available computing resources. The Generalized Likelihood Uncertainty Estimation (GLUE) technique, (Beven and Binley, 1992) – known as an informal Bayesian approaches – is one of the most widespread methods in analyzing the parameter uncertainty in hydrology and is widely used to analyze and estimate predictive uncertainty (e.g. Freer et al., 1996; Beven and Freer, 2001; Montanari, 2005). GLUE defines a cutoff threshold for its likelihood function to separate good solutions (behavioral parameters) from non-behavioral parameter sets. GLUE has

many advantages such as flexibility, ease of imple

mentation, and suitability for applying in a number of

simple to complex search problems.

GLUE has been used in many studies in hydrology

(e.g. Vrugt et al., 2008; Vrugt et al., 2009; Ste

dinger et al. 2008, McMillan and Clark, 2009). But

its flexibility on The Hydrologic Modeling System

(HEC-HMS; hereafter HMS) semi-distributed hydrol

ogy model was rarely studied before (Nourali et al.,

unpubl.). The main objective of this paper is to con

struct MC based uncertainty assessment of HEC-HMS

model to simulate streamflow over a highly complex

and heterogeneous watershed in the southeastern USA.

2 METHODOLOGY

2.1 Study area

The Waccamaw River watershed, located in the south

east coastal region of USA (North and South Carolina),

was selected for this research (Fig 1). The study area is

1886 km 2 (delineated by HEC-HMS model). The cli

mate of the region is specified as humid subtropical

and precipitation in the summer is dominated by con

vection storms; in the winter by frontal boundaries.

The long-term annual average temperature (1946–

2009) is 16.88 ∘ C, and averaged annual precipitation is a little more than 1300 mm in the same period. During the calibration period (2003–2005), precipitation ranged from an extremely wet year in 2003 (320 mm above average rainfall) to an average range (around 1350 mm).

Major soil type of this landscape is sandy loam with moderate permeability.The portion of low storage shallow soil is around 90% which is restricted in moderately well drained (B soil hydrologic group) to poorly drained soil (D soil hydrologic group) with agriculture, rangeland, forested-rangeland and forested wetland covers.

We acquired meteorological data (daily precipitation, maximum and minimum air temperatures, wind speed, humidity, and solar radiation), and spatial data inputs (digital elevation model (DEM), land use, and soil coverage) from the National Climatic Data Center (NCDC) and USGS portals, respectively. Climate data was collected for three weather stations namely Whiteville7, Longwood, and Loris, located inside the watershed boundary (see Figure 1).

2.2 HEC-HMS model

The Hydrologic Modeling System which is designed to simulate the complete hydrologic processes is widely used as a standard and versatile model for hydrologic

simulation. Many traditional hydrologic analysis pro

cedures such as event infiltration, unit hydrographs,

and hydrologic routing are included to the model.

HEC-HMS also includes procedures necessary for

continuous simulation including evapo-transpiration,

snowmelt, and soil moisture accounting. HMS uses Figure 1. Location map of the study area. two different procedures for rainfall-runoff modeling; they are the Soil Conservation Service curve number method (SCS-CN; USDA 1986) and the soil moisture accounting (SMA) model. SCS-CN is a procedure for simulating direct surface runoff from a storm event (i.e., event rainfall-runoff model) while SMA model has been incorporated in HMS to facilitate continuous hydrologic modeling. SMA is a lumped bucket-type model that represents a subbasin with well-linked storage layers/buckets accounting for canopy interception, infiltration, surface depression storage, evapotranspiration, as well as soil water and groundwater percolation. In this study, the SMA model was used for continuous hydrologic modeling. Given precipitation and potential ET, the SMA model computes basin surface runoff, groundwater flow, losses due to ET, and deep percolation over the entire basin. The schematic map of SMA method is presented in Figure 2. SMA model simulates the movement of water through, and storage of water on or in the surface and groundwater layers. The SMA loss method uses three layers to represent the dynamics of water movement in the soil. Layers include soil storage, upper groundwater and lower groundwater. The soil storage layer is subdivided into tension and gravity storage. Groundwater layers are intended to represent shallow interflow processes. 2.3 GLUE framework The generalized likelihood uncertainty procedure is a Monte Carlo method, the objective of which is to identify a set of behavioral model within the universe of possible model/parameter combination probability (Blasone et al., 2008). Based on the concept of equifinality proposed by Beven and Binley, (1992), large number of runs is initially performed with different parameter combinations chosen randomly from prior parameter distribution. By comparing predicted

Figure 2. SMA loss method schematic (HEC, 2000).

and observed responses, each set of parameter values is assigned to a likelihood value, i.e. a function that quantifies how well a particular parameter (or model) simulates the system. A higher value of the likeli hood function determines better agreement between the model prediction and observation. Based on a cut off threshold (an inflexion value of 1–2% that defines the effect of the acceptable sample rate is minor, Lu et al., 2014), the total sample of simulation is split into behavioral and non-behavioral parameter combinations. In this study, top 1% of the total sam ples were retained as behavioral sets (Lu et al., 2014; Hamraz et al., 2015), and the rest of parameter sets were considered as non-behavioral solution. The like lihood values of the retained solutions were then rescaled to obtain the cumulative distribution function (CDF) of the output prediction.

2.4 Model evaluation

In this research, three indices were used to quantify the goodness of calibration/uncertainty performance: a P-factor (Equation 1), which is the percentage of data bracketed by a 95% prediction uncertainty band (95PPU; maximum value is 100%), and a R-factor (or d-factor; Equation 2), which is the average width of the uncertainty band divided by the standard deviation of the corresponding measured variable (minimum value

is zero; Abbaspour et al., 2004; 2007). Theoretically, the value for P-factor ranges between 0 and 100%, while the R-factor ranges between 0 and infinity. Where d x is the average distance between the upper and the lower 95PPU, X U and X L represent the upper and the lower boundaries of the 95PPU, while σ x is the standard deviation of the measured data and N is number of observation data. The Nash-Sutcliffe efficiency (Equation 3) as last criteria is a normalized statistic that determines the relative magnitudes of the residual variance ("noise") compared to the measured data variance (Nash and Sutcliffe, 1970). NSE ranges from −∞ to 1 where a high value indicates a good agreement between simulated and observed streamflow. Where, Q i,observed and Q i,simulated are the observed and simulated streamflow. Q − observed represents the average of observed streamflow during calibration period. 3 RESULTS AND DISCUSSION 3.1 Parameter sensitivity The calibration was conducted during 2003–2005 with a 3year as a warm up period (i.e. 2000–2002). This study first examined 37 hydrologic parameters for the entire watershed and performed 200,000 runs, but the performance of modeling showed satisfactory results (NSE = 0.57). The research was then preceded using 18 parameters with 200,000 runs considering merely upstream outlet for simulation (downstream outlet was ignored). Analysis suggests that HMS performances improved significantly (NSE > 0.65) suggesting a large control of groundwater properties on streamflow simulation. Figures 3 to 5 summarize posterior distributions for all 18 model parameters with best values recognized by blue symbol (×). It can be resulted clearly storage coefficient parameter fairly well identified which affects mostly groundwater component. Also parameter represents maximum percolation reached the lower bound and may be reasoned by low permeability of watershed with coastal plain characteristics. Thus, it can be argued that groundwater property especially maximum percolation and storage coefficient are the Table 1. HMS calibrated parameters and their optimal values in GLUE during calibration period. Minimum Maximum Parameter Parameter Value Parameter Value Best Parameters Soil percent 0 100 96.46 GW1 percent 0 100 9.92 GW2 percent 0 100 47.47 Maximum Infiltration 0 20 14.74 Storage capacity 0 15 2.83 Tension capacity (multiplier) 0 1 0.28 Maximum percolation 0 20 1.01 GW1 storage capacity 0 25 2.55 GW1 routing coefficient 1 1000 234.50 GW1 maximum percolation 0 10 3.57 GW2 storage capacity 0 30 19.52 GW2 routing coefficient 1 1500 1088.17 GW2 maximum percolation 0 15 13.79 Time of concentration 2 300 56.07 Storage coefficient 400 800

628.04 Recession factor 0 1 0.51 Initial baseflow 0 10 3.06 Threshold to peak flow 0 1 0.33

Figure 3. Marginal posterior probability distributions of HMS parameters inferred using GLUE.

most sensitive parameters. In HMS modeling, tension capacity should be less than storage capacity; there fore a multi-value for the tension capacity is used in this study.

Apart from most of parameters which have uniform and or multi-peaked distributions slightly, GW1 rout ing coefficient and GW2 routing coefficient parame ters are distributed uniformly well and show minimum level of sensitivity and could be assumed a constant value so they were ignored from calibration period. 3.2 Calibration result Calibration modeling showed that HMS simulated streamflow well (Fig. 6) during calibration period (three years as 1096 days). While in some intervals, measured values are extrapolated out of the 95PPU bound, the 95PPU is quite suitable to bracket the observations in 2004 and 2005, while it is slightly underestimated peak flows in 2003 due to extreme wet condition (higher than average annual rainfall)

Figure 4. Marginal posterior probability distributions of HMS parameters inferred using GLUE.

Figure 5. Marginal posterior probability distributions of HMS parameters inferred using GLUE.

in the watershed. Extremely wet hydrological condi tion increase the complexities in river regulation and hydraulic characteristics at the Waccamaw watershed which in turn have a large influences on groundwa ter properties. Therefore HMS showed less skill to capture repeated wet condition in the coastal plain

watershed. Overall, the calibration results showed a

skillful calibration by presenting P-factor = 60.7%

and R-Factor = 1.02. 4 CONCLUSIONS This study coupled HMS model with GLUE uncertainty algorithm for daily streamflow simulation of a coastal plain watershed. Calibration was conducted for each outlet individually and for the entire watershed as well. Analysis suggests HMS showed less skill in downstream portion with high groundwater contribution while showed relatively better performance

Figure 6. Daily streamflow simulation during calibration period at the Freeland station.This outlet showed better performance

compare to downstream outlet.

at upstream gauge. Overall, HMS could skillfully

simulate the magnitudes and frequency of moderate

streamflow while showed deficiency in extreme sim

ulation (both high and low flow events). Examining

more accurate climate data (radar data) and satel

lite images are highly recommended for taking into

account the climatic, hydrological and soil spatial

variability in this heterogeneous watershed.

ACKNOWLEDGEMENT

This research was partially supported by the University

of South Carolina (SCFLOODS grant: # 15520-16

40787).

Abbaspour, K. C., Johnson, C. A., & Van Genuchten, M. T.

2004. Estimating uncertain flow and transport parameters

using a sequential uncertainty fitting procedure. Vadose

Zone Journal, 3(4), 1340-1352.

Abbaspour, Karim C., Jing Yang, Ivan Maximov, Rosi Siber,

Konrad Bogner, Johanna Mieleitner, Juerg Zobrist, &

Raghavan Srinivasan, 2007 Modelling hydrology and

water quality in the pre-alpine/alpine Thur watershed

using SWAT. Journal of Hydrology 333: 413-430.

Beven, K., & Freer, J. 2001. Equifinality, data assimilation,

and uncertainty estimation in mechanistic modelling of

complex environmental systems using the GLUE method

ology. Journal of hydrology, 249(1), 11-29.

Beven, K., & Binley, A. 1992. The future of distributed

models: model calibration and uncertainty prediction.

Hydrological processes, 6(1), 279-298.

Blasone, R. S., Vrugt, J. A., Madsen, H., Rosbjerg, D.,

Robinson, B. A., & Zyvoloski, G. A. 2008. Generalized

likelihood uncertainty estimation (GLUE) using adaptive Markov Chain Monte Carlo sampling. Advances in Water Resources, 31(4), 630-648. Boyle DP, Gupta HV, Sorooshian S, 2000 Toward improved calibration of hydrological models: combining the strengths of manual and automatic methods. Water Resour Res;36:3663-74. Duan, Q., Sorooshian, S., & Gupta, V. K. 1994. Optimal use of the SCE-UA global optimization method for calibrating watershed models. Journal of hydrology, 158(1), 265-284. Freer J, Beven K, Ambroise B 1996 Bayesian estimation of uncertainty in runoff prediction and the value of data: an application of the GLUE approach. Water Resour Res 32(7): 2161-2173. Gan, T. Y., & Biftu, G. F. 1996. Automatic calibration of conceptual rainfallrunoff models: Optimization algorithms, catchment conditions, and model structure. Water resources research, 32(12), 3513-3524. Hamraz, B., Akbarpour, A., Bilondi, M. P., & Tabas, S. S. 2015. On the assessment of ground water parameter uncertainty over an arid aquifer. Arabian Journal of Geosciences, 8 (12), 10759-10773.

Hydrologic Engineering Center (HEC). 2000. Hydrologic Modeling System, HEC-HMS. Technical Reference Manual. U.S.Army Corps of Engineers, Davis, California. Janssen, P. H. M., & Heuberger, P. S. C. 1995. Calibration of process-oriented models. Ecological Modelling, 83(1), 55–66. Lu D., Ye Ming, Hill M.C., Poeter E.P., Curtis G.P. 2014. A computer program for uncertainty analysis integrating regression and Bayesian methods, Environmental Modelling & Software, 60:45–56. McMillan, H., & Clark, M. 2009. Rainfall-runoff model calibration using informal likelihood measures within a Markov chain Monte Carlo sampling scheme. Water Resources Research, 45(4). Montanari, A. 2005. Large sample behaviors of the generalized likelihood uncertainty estimation (GLUE) in assessing the uncertainty of rainfall-runoff simulations. Water Resources Research, 41(8).

Nash, J., & Sutcliffe, J. V. 1970. River flow forecast ing through conceptual models part I—A discussion of principles. Journal of hydrology, 10(1), 282–290.

Nourali, M., Ghahraman, B., Pourreza-Bilondi,M. & Davari, K, Choice of likelihood function for estimating uncer tainty of HEC-HMS Flood Simulation Model Using Markov Chain Monte CarloAlgorithm, Journal of Hydrol ogy, in prep.

Stedinger, J. R., Vogel, R. M., Lee, S. U., & Batchelder, R. 2008. Appraisal of the generalized likelihood uncertainty estimation (GLUE) method. Water resources research, 44(12).

Uhlenbrook, S., Seibert, J., Leibundgut, C., & Rodhe, A. 1999. Prediction uncertainty of conceptual rainfall-runoff models caused by problems in identifying model parame ters and structure. Hydrological Sciences Journal, 44(5), 779–797.

USDA, S. 1986. Urban hydrology for small watersheds.

Technical release, 55, 2–6.

Vrugt JA, Gupta HV, Bastidas LA, Bouten W, Sorooshian

S 2003a. Effective and efficient algorithm for multi

objective optimization of hydrologic models. Water

Resour Res; 39(8):1214. Vrugt JA, Gupta HV, Bouten W, Sorooshian S. 2003b, A Shuffled Complex Evolution Metropolis algorithm for optimization and uncertainty assessment of hydrologic model parameters. Water Resour Res;39(8):1201. Vrugt, J. A., Ter Braak, C. J., Gupta, H. V., & Robinson, B.A. 2009. Equifinality of formal (DREAM) and informal (GLUE) Bayesian approaches in hydrologic modeling? Stochastic Environmental Research and RiskAssessment, 23(7), 1011–1026. Vrugt, J.A., ter Braak, C.J.F., Clark, M.P., Hyman, J.M., Robinson, B.A., 2008. Treatment of input uncertainty in hydrologic modeling: Doing hydrology backward with Markov chain Monte Carlo simulation.Water Resour. Res. 44(12), W00B09. Yapo PO, Gupta HV, Sorooshian S, 1998 Multi-objective optimization for hydrologic models. J Hydrol;204:83–97. Yapo, P. O., Gupta, H. V., & Sorooshian, S. 1996. Automatic calibration of conceptual rainfall-runoff models: sensitivity to calibration data. Journal of Hydrology, 181(1), 23–48.
Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

A methodology to account for rainfall uncertainty at the event scale in fully distributed rainfall runoff models

I. Fraga, L. Cea, J. Puertas & M. Álvarez-Enjo

Environmental and Water Engineering Group, Departamento de Métodos Matematicos y de Representación,

University of A Coruña, Spain

S. Salsón & A. Petazzi

MeteoGalicia, Xunta de Galicia, Spain

ABSTRACT: The increasing importance of hydrological models as management and prediction tools has

triggered the need of quantifying the uncertainty of their predictions. These uncertainties result from multiple

factors. In this paper we present a new methodology to account for rainfall uncertainty. The methodology is

based on adding an error function to the rainfall prediction in every point. This error function is determined from

cross-validation analysis of the available rain gauge data. The error functions are then sampled using random

fields. The proposed methodology is firstly validated using rain data from 7 rain events. Then, a fully distributed

hydrological model based on the 2D shallow water equations is then used to simulate an additional rain event in

a 24 km 2 catchment, taking into account the rainfall prediction uncertainties and quantifying their effect on the

computed discharge at the catchment outlet.

1 INTRODUCTION

Quantifying the uncertainty on the discharge predic

tions obtained with hydrological models has become a

major concern in the last decades. Model output uncer

tainty stems from several factors as observation errors

in the data used for calibration, uncertainty of ini

tial conditions and input data, parameter estimation,

and mathematical simplifications. Among the differ

ent inputs used by these hydrological models, rainfall

has been pointed as the one with higher uncertainties

and one of the main sources of errors in model predic

tions (Bardossy & Das 2008, Renard et al. 2011among others).

The uncertainty in rainfall prediction is usually quantified either by stochastically perturbing the pre cipitation inputs or through conditional simulation methods. In the stochastic perturbation approach the model rainfall inputs are perturbed with an error function computed based on order of magnitude con siderations (Turner et al. 2008, Pan & Wood 2009, Eliézer et al. 2015) or they include precipitation multi pliers to represent systematic errors in rainfall forcing data which are calibrated in conjunction with all the model parameters (Kavetski et al. 2006, Vrugt et al. 2008). However, uncertainty in precipitation estimates tends to vary both spatially and temporally (Tian & Peters-Lidard 2010, Sorooshianet al. 2011), and esti mates of precipitation uncertainty from such order-of magnitude or systematic-error based approaches may be statistically unreliable as the spatial variability is ignored. One possible way of overcoming this drawback is using conditional simulations. Conditional simulations is a geostatistical method that generates multiple replicates of a rainfall field, each of them representing an equiprobable rainfall event (e.g. Vischel et al. 2009). This method has the advantage of introducing the spatial variability on the uncertainty estimation without many assumptions on the rainfall error distribution (Clark & Slater 2006). This approach has been successfully addressed by several authors to quantify precipitation uncertainty in rainfall-runoff predictions (Götzinger & Bárdossy 2008, AghaKouchak et al. 2010, Renard et al. 2011). However, conditional simulation methods can be data-intensive to parameterize and computationally expensive to run (McMillan et al. 2011) which limits its application. The proposed methodology combines both approaches. On the one hand, it follows the stochastically perturbing approach by adding an error function to the rainfall prediction in every point. On the other hand, the obtained error functions are sampled taking into account the spatial correlation, following the geostatistical approach of the conditional simulations. 2 METHODOLOGY DESCRIPTION This methodology considers that the predicted rainfall results from the sum of two components: the interpolated rainfall and the prediction uncertainties. To compute the interpolated rainfall first an isotropic climatological variogram is inferred from the rain gauges data. Then, this variogram is used to interpolate the rainfall in every point of the domain using the ordinary kriging technique.

The prediction uncertainties correspond to multiple equi-probable error fields, which are estimated according to the following procedure which is repeated each time step. Firstly an error function is defined for each point of the domain. In the proposed methodology it is assumed that the error function in every point follows a normal distribution.The mean value of the error is considered zero in the whole domain, while the standard deviation of the error varies in space and time, and it is determined from cross-validation computations according the procedure described in Delrieu et al. (2014). After the standard deviation of the error field is computed, the error functions are sampled. To do so, multiple non-conditioned Gaussian random fields are generated and the errors in each point corresponds to the product of the standard deviation multiplied by the values of the Gaussian fields. In the computations presented in further sections of this paper, 100 Gaus sian fields were generated for all the time steps of each rain event. This number of simulated fields was suffi cient to estimate rainfall predictions uncertainties with a reasonable computational burden.

Gaussian random fields are generated using an spherical variogram with zero nugget and a sill equal to one. Regarding the range of the variogram, follow ing Severino & Alpuim (2005) since each gauge is a different device measurement errors are assumed to be mutually uncorrelated on space. Therefore, the errors between any two rain gauges are considered indepen dent and the variogram range is set equal to the lowest distance between rain gauges. Imposing a sill of one ensures that the random numbers generated in each point follow a normal distribution with zero mean and unitary standard deviation.

3 METHODOLOGY VALIDATION

3.1 Study area

The described methodology was used to predict the rainfall fields and account for prediction uncertainties in a study region which covers an area of 60x60 km of the east coast of Galicia, in the Northwest of Spain

(Fig. 1). Due to its location at the junction between two climatic areas, the study area is affected by the ener getic interchanges between tropical and polar regions. This results in frequent adverse weather phenomena, inherent to the Galician mild oceanic climate, consist ing of strong winds and intense rainfalls characterized by a great variability in terms of both space and time (Cabalar Fuentes 2005).

The Hortonian processes are the key factors involved in the generation of the surface runoff in the catchments within the region. This fact, in addition to the variability of the rainfall previously referred, high lights the importance of an accurate characterization of the rainfall during a rain event towards a further use of the meteorological data as inputs in hydrological models. Figure 1. Caption of the study area and the con Catchment, including the MeteoGalicia (MG), CSIC and CAPRI rain gauges. UTM coordinates referred to ETRS89 datum. In order to monitor this highly variable rain phenomenon, Galicia has a dense rain gauge network managed and operated by the regional weather agency MeteoGalicia. 17 rain gauges from this network and one additional rain gauge operated by the CSIC (Consejo Superior de Investigaciones Científicas, which belongs to the Spanish Ministry of Economy and Competitiveness) were identified inside the study area. All these rain gauges are tipping-bucket pluviometers with a resolution of 0.2 mm and a measuring frequency 10 min. Three additional rain gauges with the same characteristics as the ones operated by MeteoGalicia and the CSIC were installed in the river Con catchment (Fig. 1).These gauges were installed within the CAPRI project, financially supported by the Spanish Ministry of Economy and Competitiveness (Ministerio de Economía y Competitividad). 3.2 Validation rain events 7 events registered during the year 2015 were selected to validate the proposed methodologies. The characteristics of the events are summarized in Table 1, where the rainfall temporal variability was computed as the ratio between the standard deviation and the mean value of the rain intensity during the rain event. An additional rain event (E08) was used in later rainfall-runoff simulations of the Con catchment (see section 4). As pointed by several authors (Ciach 2003, Molini et al. 2005, Villarini et al. 2008), the accuracy of the tipping-bucket rain gauges decreases with decreasing accumulation times, and the use of these devices for temporal scales lower than 10–15 minutes is usually inappropriate (Habib et al. 2001). To take this onto account, the rainfall depths measured every 10 minutes were aggregated into 30 minutes intervals.

Table 1. Characteristics of the rain events at the CSIC network rain gauge.

Event	Duration (h)	Rainfall depth (mm)	Peak Rain intensity (mm/h)	Rainfall temporal variability
E1	5	15.4	16.8	1.45
E2	21	61.2	36.0	1.72
E3	11	25.0	19.2	1.56
E4	7	26.6	55.2	2.54
E5	10	18.4	14.4	1.48
E6	14	32.6	9.6	0.88
E7	14	13.6	4.8	1.07
E8	23	50.4	9.4	1.11

Table 2. Mean NSE indices and percentages of data coverage of the validation rain events in the 3 validation rain gauges.

Event	E1	E2	E3	E4	E5	E6	E7
NSE	0.89	0.88	0.88	0.92	0.74	0.57	0.81
Cover	1.00	0.99	0.95	1.00	0.94	0.81	0.96

Experimental data of the events E1–E7 from the MeteoGalicia and CSIC networks were used to predict the rainfall and to account the prediction uncertain ties. Those predictions were then validated using the rainfall data from the CAPRI rain gauge stations.

3.3 Validation results

Results show that the mean hyetographs are cor rectly predicted with the proposed methodology. The computed hyetographs present a good fit to the exper imental data with Nash-Sutcliffe Efficiency indices higher than 0.7 in all the events except one. In terms of the prediction uncertainty, the experimental data within the 95% confidence intervals exceeds the 80% in all the events (Table 2). These coverage values in model validation are common in the application of the uncertainty prediction in hydrological modeling (Vrugt et al. 2009).Also, most of the experimental val ues outside the confidence bounds occur during time periods of low rainfall depths, where the inaccuracy of the measuring devices deteriorates

The amplitude of the uncertainty bounds is par ticularly sensitive to the rain intensity and higher uncertainties are expected in the rain events with higher rainfall intensity. However, this increase of the uncertainty bounds also results in higher coverage of the experimental data (Fig. 2).

4 RAINFALL RUNOFF MODELLING

INCLUDING RIANFALL UNCERTAINTIES

4.1 Objectives and scope

Once the methodology to account for rainfall pre

diction uncertainties was validated, we analyzed how Figure 2. Caption of the maximum rain intensity versus uncertainty bound amplitude (a) and bound amplitude versus data coverage (b). these uncertainties affect the predicted hydrograph at a catchment outlet.To do so, the predicted rainfall fields, including the prediction uncertainties, were used as inputs in an hydrological fully distributed 2D model of the river Con catchment. 4.2 Hydrological model The hydrological model used in the computations presented in this section consists of a distributed groundwater flow model implemented in the existing surface runoff model Iber (Bladé et al. 2014). The Iber model solves the 2D shallow-water equations, including rainfall and infiltration terms, with an explicit unstructured finite volume solver. The model has been validated in previous studies under overland flow conditions including rainfall-runoff transformation (Cea & Bladé 2015, Cea et al. 2014). Infiltration losses have been estimated using a simple model that considers an initial rainfall abstraction and a constant infiltration rate. The initial abstraction models the rain losses due to the interception by the vegetation and the initial infiltration losses until the soil gets saturated. After the soil is fully saturated the infiltration capacity is reduced to a constant value. The underground flow was computed solving the Boussinesq 2D equation for unconfined flow in a porous isotropic media, following the Dupuit hypothesis. In order to take into account the interaction between surface and groundwater flows, in the referred equation the recharge due to the infiltration and the discharge to the exfiltration to the surface are included. This model does not consider the flow at the nonsaturated zone, and the infiltrated water is directly conveyed into the saturated zone, contributing to the increase of the water table. This approach is only acceptable for small soil depths and high vertical conductivities. The underground flow equation is solved with an explicit unstructured finite volume solver using the same mesh as for the surface runoff flow. 4.3 Catchment description and model The river Con is located in the Northwest of Spain, in the province of Pontevedra. Despite the relatively small catchment area (24 km 2), the elevations within the catchment range from 640 meters above the sea level (at the Northeast of the catchment) to the sea

Figure 3. Caption of the Con catchment land uses, pressure gauge and CAPRI rain gauge network locations.

level at the west side of the catchment, where the river Con discharges into the Arousa estuary after crossing Vilagarcia de Arousa. Vilagarcia de Arousa, of approximately 30000 inhabitants, suffers frequent floods which have significant economical costs due to the damages caused to infrastructures and private properties.

The catchment was discretized in an unstructured mesh of 57252 triangular elements. The size of the elements varies from 10 m. at the river Con and its tributaries to 35 m. at the rest of the catchment. 3 types of land uses (urban, agricultural and forest) were identified within the catchment (Fig. 3).

4.4 Model calibration and discharge uncertainty quantification

Prior to the estimation of the uncertainty of the dis charge computed at the catchment outlet, the surface Manning coefficient, the infiltration rates, the width of the permeable soil and the hydraulic conductivity of the hydrological model were calibrated using rainfall and discharge data of the event E8 (Cea et al. 2015).

The rainfall was interpolated from rain data from all the rain gauge networks using the ordinary krigging technique. This rain data was assumed to be error free. Discharge data was obtained from the water level measured with a pressure gauge was installed close to the river Con outlet (Fig. 3).

Once the model was calibrated, the same event (E8) was simulated including the uncertainty of the predicted rainfall computed following the proposed methodology.

From the computed discharges, confidence bounds of the discharge at the catchment outlet were deter mined (Fig. 4). Results show that the shape of the hydrograph is correctly captured with the hydrologi

cal model. A better performance of the hydrological Figure 4. Caption of the computed discharges and predicted rainfall confidence bounds. model is noticed at the second peak of the hydrograph, which takes place around 21 hours after the begging of the rain event. The measured discharges during the referred peak are within the 95% confidence interval. On the contrary, the first peak of the hydrograph is underestimated and the measured discharges lie outside the confidence bounds. It is also interesting to notice the significant effect of the rainfall uncertainty bounds of the computed hydrograph. During the second peak of the hydrograph, the bound amplitude of the predicted discharge is nearly 0.6 m 3 /s, which represents around 50% of the peak discharge.Therefore, the uncertainty on discharge predictions ascribable to the uncertainty of the rainfall predictions becomes significant in the studied case. 5 CONCLUSIONS A new methodology to account for rainfall uncertainty at the event scale was presented. Following this methodology, the uncertainty of the predicted rainfall is quantified by defining an error function in each point of the studied area, which is subsequently sampled.The referred error function is a zero-mean normal distribution. The standard deviation of the distribution is computed from cross-validation analysis of the available rain gauge data. In order to sample the error functions, multiple Gaussian random fields are generated, and the errors in the rainfall prediction are computed as the product between the random fields and the standard deviation field. The proposed methodology was validated analyzing 7 rain events which occurred at the Northwest of Spain during the year 2015. Rain data from MeteoGalicia and CSIS rain gauge networks were used to quantify rainfall uncertainties following the proposed methodology, which were then validated using rain data from 3 additional stations. Results showed a good performance of the proposed methodology, and the uncertainty bounds of the rainfall predictions covered more than 80% of the experimental data in all the events. A rain event, different to the ones used to validate the methodology, was used to quantify the effect of rainfall uncertainties on the discharges computed by a

distributed hydrological model in a small catchment.

To do so, the hydrological model was used to simulate the referred rain event including the rainfall prediction uncertainties on the model inputs. The effect of the rainfall uncertainty on the discharge turned out to be significant, specially for higher discharges. During the peak of the hydrograph the amplitude of the confidence bounds was half of the mean computed discharge.

6 FUTURE WORK

Simulations of a single rain event using a previously calibrated fully distributed hydrological model showed the impact of the rainfall uncertainty on model predictions. This highlighted the importance of including uncertainty quantifications in all the stages of the rainfall-runoff modeling. Therefore, in future works the uncertainty of the predicted rainfall must be included in the calibration of the hydrological model.

AghaKouchak, A., Bárdossy, A., & Habib, E. (2010). Con ditional simulation of remotely sensed rainfall data using a non-Gaussian v-transformed copula. Advances in water resources 33(6): 624–634.

Bárdossy,A. & Das,T. 2008. Influence of rainfall observation network on model calibration and application. Hydrology and Earth System Sciences Discussions 12(1): 77–89.

Bladé, E., Cea, L., Corestein, G., Escolano, E., Puertas, J.,

Vázquez-Cendón, E., & Coll, J. 2014. Iber: herramienta de simulación numérica del flujo en ríos. Revista Interna cional de Métodos Numéricos para Cálculo y Diseño en Ingeniería 30(1): 1–10.

Cabalar Fuentes, M. 2005. Los temporales de lluvia y viento en Galicia. Propuesta de clasificación y análisis de ten dencias (1961–2001). Investigaciones geográficas 36: 103–118.

Cea, L., Legout, C., Darboux, F., Esteves, M., & Nord, G. 2014. Experimental validation of a 2D overland flow model using high resolution water depth and velocity data. Journal of Hydrology 513: 142–153.

Cea, L. & Bladé, E. 2015. A simple and efficient unstructured finite volume scheme for solving the shal low water equations in overland flow applications.

Water Resources Research Accepted Author Manuscript.

doi:10.1002/2014WR016547.

Cea, L., Fraga, I., Puertas, J., Álvarez, M., Bermúdez, M., Coquerez, S., Salsón, S. & Petazzi, A. 2015. Influencia de la densidad espacial de estaciones pluviométricas y de la disponibilidad de datos radar en los hidrogramas de tor menta calculados con un modelo hidrológico distribuido: Aplicación a una cuenca de 24 Km2 en el Noroeste de España. Proceedings of the IV Jornadas de Ingeniería del Agua, Córdoba (Spain).

Clark, M.P. & Slater, A.G. 2006. Probabilistic quantitative precipitation estimation in complex terrain. Journal of Hydrometeorology 7: 3–22.

Ciach, G.J. 2003. Local random errors in tipping-bucket rain gauge measurements. Journal of Atmospheric and Oceanic Technology 20(5): 752–759.

Delrieu, G., Wijbrans, A., Boudevillain, B., Faure, D., Bon nifait, L. & Kirstetter, P.E. 2014. Geostatistical radar-raingauge merging: A novel method for the quantification of rain estimation accuracy. Advances in Water Resources 71: 110–124. Eliézer, B.I., Charlene, G., Eric, A. A., & Abel, A. 2015. Influence of the uncertainties related to the Random Component of Rainfall Inflow in the Ouémé River Basin (Benin, West Africa). International International Journal of Current Engineering and Technology 5(3): 1618–1629. Götzinger, J., & Bárdossy, A. 2008. Generic error model for calibration and uncertainty estimation of hydrological models. Water resources research 44(12). Habib, E., Krajewski, W.F., & Kruger, A. 2001. Sampling errors of tipping-bucket rain gauge measurements. Journal of Hydrologic Engineering 6(2): 159–166. Kavetski, D., Kuczera, G. & Franks S.W. 2006. Bayesian analysis of input uncertainty in hydrological modelling: 1. Application. Water Resources Research 42(3). Molini, A., Lanza, L.G. & La Barbera, P. 2005. The impact of tipping-bucket raingauge measurement errors on design rainfall for urban-scale applications. Hydrological processes 19(5): 1073–1088. McMillan, H., Jackson, B., Clark, M., Kavetski, D. & Woods, R. 2011. Rainfall uncertainty in hydrological modelling: An evaluation of multiplicative error models. Journal of Hydrology 400: 83–94. Pan, M. & Wood, E.F. 2009. A multiscale ensemble filtering system for hydrologic data assimilation. Part II: Application to land surface modeling with satellite rainfall forcing. Journal of Hydrometeorology 10(6): 1493–1506. Renard, B., Kavetski, D., Leblois, E., Thyer, M., Kuczera, G., & Franks, S. W. 2011. Toward a reliable decomposition of predictive uncertainty in hydrological modeling: Characterizing rainfall errors using conditional simulation. Water Resources Research 47(11). Severino, E., & Alpuim, T. 2005. Spatiotemporal models in the estimation of area precipitation. Environmetrics 16(8): 773–802. Sorooshian, S., AghaKouchak, A., Arkin, P., Eylander, J., Foufoula-Georgiou, E., Harmon, R., Hendrickx, J.M.H., Imam,B., Kuligowski, R., Skahill, B. & SkofronockJackson, G. 2011. Advanced concepts on remote sensing of precipitation at multiple scales. Bulletin of theAmerican Meteorological Society 92(10): 1353–1357. Tian, Y. & Peters-Lidard, C.D. 2010 A global map of uncertainties in satellite based precipitation measurements. Geophysical Research Letters 37(24). Turner, M.R.J., Walker, J.P., & Oke, P.R. 2008. Ensemble member generation for sequential data assimilation. Remote Sensing of Environment 112(4): 1421–1433. Villarini, G., Mandapaka, P.V., Krajewski, W.F., & Moore, R.J. 2008b. Rainfall and sampling uncertainties: A rain gauge perspective. Journal of Geophysical Research: Atmospheres 113(D11). Vischel, T., Lebel, T., Massuel, S. & Cappelaere, B. 2009. Conditional simulation schemes of rain fields and their application to rainfall-runoff modeling studies in the Sahel. Journal of Hydrology 375 (1–2): 273–286. Vrugt, J.A., ter Braak, C.J.F., Clark, M.P., Hyman, J.M. & Robinson B.A. 2008. Treatment of input uncertainty in hydrologic modeling: doing hydrology backwards with Markov Chain Monte Carlo simulation Water Resources Research 44(12). Vrugt J.A., Ter Braak C.J., Gupta H.V., Robinson B.A. 2009. Equifinality of formal (DREAM) and informal (GLUE) bayesian approaches in hydrologic modeling. Stochastic environmental research and risk assessment 23 (7): 1011–1026. Sustainable Hydraulics

in the Era of Global Change – Erpicum et al. (Eds.) © 2016
Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Estimating probability of dike failure by means of a Monte Carlo approach

R. Van Looveren, T. Goormans & J. Blanckaert

International Marine and Dredging Consultants nv, Antwerp, Belgium

Kristof Verelst & Patrik Peeters

Flanders Hydraulics Research, Antwerp, Belgium

ABSTRACT: The European Flood Directive 2007/60/EC requires Member States to assess the flood risk along their water courses and to take adequate measures to reduce this risk. Often flood risk maps are developed by only taking into account dike overtopping. However, floods can also be caused by dike failure. Not taking this into account underestimates the flood risk. This paper presents a method to estimate the probability of dike section failure, taking into account several failure mechanisms. The method follows a Monte Carlo approach, in which all relevant parameters for the hydraulic loads as well as for the considered resisting forces vary according to specific distributions. Both the loads and failure mechanisms are described using numerical models. This method has been successfully applied on various dike segments along navigable water courses in Flanders, and in the Periodic Safety Review of the nuclear power plant of Doel.

1 INTRODUCTION

The European Flood Directive 2007/60/EC requires Member States to assess the flood risk along their water courses and coast lines, to map the flood extent

together with assets and humans at risk in these areas, and to take adequate measures to reduce this risk.
Often flood risk maps are developed, in which the probability of the flood extent is quantified by only taking into account dike overtopping. However, struc tural dike failure according to different mechanisms can also cause floods. Not taking these mechanisms into account underestimates the flood risk.
This paper presents a method to estimate the prob ability of failure of a dike section, taking into account several failure mechanisms. The method follows a Monte Carlo approach, in which all relevant parame ters for the hydraulic loads as well as for the considered resisting forces vary according to specific distribu tions. Besides these structural parameters, also the loads vary according specific distributions. Both the loads and failure mechanisms are described using physically-based numerical models.
The method has been applied to various dike seg ments along navigable water courses in Flanders, and in the Periodic Safety Review of the nuclear power plant of Doel along the Scheldt estuary. Results of the method can be used in impact analyses and flood risk mapping.
This paper first describes the methodology, then

gives some practical examples on the implementation,

and ends with conclusions and recommendations. 2 THE METHODOLOGY 2.1 General The failure of a dike section is evaluated by comparing the loading forces R to the resisting forces S, both described through numerical models. The dike section fails if the load is higher than the resistance, i.e. if $R - S > 0$ (limit state equation). There is a different limit state equation for each of the five failure mechanisms considered here: macro instability (or global instability), micro instability (or local instability), piping, and erosion of the inner slope and erosion of the outer slope. Section 2.2.2 explains these failure mechanisms in more detail. To take into account the existing uncertainty on the load as well as on the resistance, each of these five limit state equations is calculated for different sets of loads and resistances. At the load side, this set is constituted by a probabilistic set of synthetic storms (see sections 3.1.3 and 3.2.3 for examples). At the resistance side, a stratified sampling technique on the resistance parameters is performed to compose the different sets. Since the probability distribution function (pdf) is known, the probability of each set can be determined. Physically-based numerical models are then used (i) to transfer the load at the boundary conditions of a region to the specific dike section and (ii) to evaluate the different failure mechanisms. By calculating each of these five limit state equation for several combinations of load sets and resistance sets (Monte Carlo approach), the full spectrum of

possible combinations can be considered, and hence

the global probability of failure can be determined.

2.2 Load

2.2.1 General

Figure 1 shows in more detail the methodology at the

load side (R). This figure is explained from bottom to

top in this and the following section.

The load at a dike section x (bottom of the figure)

is defined by the water level variation, the position of the phreatic line within the dike and the wave conditions (defined by the wave amplitude H_s and wave period T_p). Long-term measurements of all these variables, necessary for the statistical approach, are seldom available at the dike location. However, by means of (physically-based) numerical models the load at the boundaries (based on a statistical analysis of long-term measurements) available at other locations can be transferred to the dike location.

2.2.2 Used models

Water level and flow velocity at the dike segment is calculated by hydrodynamic models (1D or 2D) using the relevant boundary conditions.

The position of the phreatic line within the dike, which is of prime importance for the global and local stability, can be calculated using a finite element soft ware package, e.g. SEEP/W (Geo-Slope 2012). These calculations however are time consuming, making it impractical for the Monte Carlo approach. Therefore the methodology relies on the creation of response surfaces based on a priori SEEP/W calculations for different conditions. This way fast computation time (in the Monte Carlo simulations) is combined with rel atively high accuracy. Separate response curves were established for tidal and non-tidal rivers. For each river type the phreatic line was schematised using a limited number of points.An example for a tidal river is shown in Figure 2, using four points: one point at the inner side of the dike (4), two points at the outer side cor responding to high and low water (1, 2) and one point within the dike body indicating intrusion of the tidal influence in the dike (3). The position of the phreatic line depends mainly on the dike geometry, dike per meability and the shape and duration of the storm. These parameters were varied in numerous SEEP/W calculations, and the positions of the four points was expressed as a function of the input parameters.A sim ilar approach was used for non-tidal rivers, of course resulting in different response curves.

For the calculation of the wave load the formula of Bretschneider (TAW 1985) can be used. With this formula wave characteristics (H_s and T_p) can be deter mined from wind speed (w_s), wind direction (w_d), water depth, and fetch length. It is assumed that wind data from the nearest station are representative for the dike location as well, i.e. no model is used to trans fer the wind load from the boundary conditions to the dike location, as opposed to the hydraulic load. It is

Figure 1. Determination of the load (R). Figure 2. Schematization of the phreatic line for a tidal river.

expected that little gain in accuracy is achieved by calculating the nearby wind velocity field, compared to the extra effort. 2.2.3 Boundary conditions The statistical information in the boundary conditions is based on a multivariate frequency analysis of the principal variables – i.e. taking into account crosscorrelations when appropriate – while the secondary variables are kept constant. The distinction between

Figure 3. Determination of the resistance (S).

principal and secondary variables is based on a sensi

tivity analysis. This makes it possible to determine the

relative share of each variable in the resulting statistical

distribution.

For the tidally influenced Scheldt estuary, the wind

speed $w_s(t)$, wind direction $w_d(t)$, and water level $h(t)$

are principal variables, while the upstream discharge

$Q(t)$, the duration $d(t)$ and the storm surge steepness

$s(t)$ are of secondary importance, and can be treated

conditionally. For the more upstream rivers discharge

is the principal load variable, while the wind speed and

direction are secondary variables.

2.3 Resistance

2.3.1 General

Figure 3 shows more details on the methodology at

the resistance side (S). The value of each geotechni

cal parameter varies according to a pdf, which can

be approximated by normal or lognormal pdfs for all

considered parameters. Hence it is fully described by

a standard deviation σ and mean value μ.

When data is available the pdf is determined based on a statistical analysis. In absence of data, often the case for geotechnical variables, the pdfs are determined based on expert judgment and literature data (TNO 2003b, Kortenhaus et al. s.d.).

Since it is not possible to consider the full, continuous domain of resistance parameters, the pdf is discretised using stratified sampling. For each parameter of the considered failure mechanism a discrete number of samples is taken from its associated pdf.The probability of occurrence of a combination of resisting parameters is then calculated from the combination of

the respective probabilities of each single parameter value. The higher the number of samples the better the accuracy but also the more computation time is needed. Taking five samples was considered adequate. 2.3.2 Used models The following section gives more information on the considered failure mechanisms and the models used to describe them. A more extensive description of failure of dikes can be found in standard text books, e.g. Pilarczyck (1998). Global instability of the inner and outer slope is calculated by Bishop's method (Bishop 1955, Verruijt 1995), which requires the geotechnical parameters cohesion (c [kN/m^2]), friction angle (ϕ [$^\circ$]), wet soil weight (γ_n [kN/m^3]), dry soil weight (γ_d [kN/m^3]) and permeability (k [m/s]) as input. Since the Bishop calculations are time consuming the number of parameters can be reduced based on a sensitivity analysis. For a dike with a core of sand it appeared that only parameters ϕ, k and γ_n should be varied, for a clay core the parameters c, ϕ, and γ_d . Global instability is evaluated at both the inner and outer side. Local instability, which occurs when the phreatic pressure is too high, in turn causing the (impermeable) top layer to be pushed off, is evaluated by verifying the equilibrium of active and passive forces on the upper layer of the dike and/or the revetment. Following parameters are considered: thickness (d_{top} [m]), weight (γ_{top} [kN/m^3]), cohesion (c_{top} [kN/m^2]) and friction angle (ϕ_{top} [$^\circ$]) of the upper clay layer and thickness (d_{rev} [m]) and density (ρ_{rev} [kg/m^3]) of the revetment (if present). Local instability is evaluated at both the inner and outer side. Piping can occur when the water level difference over the dike is too large, causing a flow underneath the dike that erodes the particles of the base material. Sellmeyer's method (Sellmeyer 1988) is used for evaluating piping, based on the following parameters: thickness (d_{top} [m]) and specific weight (γ_{top} [kN/m^3]) of the upper clay layer, sand aquifer thickness (D_s [m]), permeability (k_s [m/s]) and specific weight (γ_s [kN/m^3]) of the porous material, and grain diameter exceeded by 30% of the grains (d_{70} [m]). The rolling resistance angle (θ [$^\circ$]) and White's constant (η [–]) are taken from Sellmeyer (1988). Water flowing along the dike, or run-up of (windinduced) waves can cause erosion of the outer slope. The resistance, and the corresponding limit state equation, depend on the revetment, e.g. grass, rip rap or a combination of both (rip rap at the bottom side of the dike and grass at the higher side). The limit state equation for rip rap is based on Pilarczyk's formula and uses the d_{50} [m] of the rip rap as main parameter (Pilarczyk 1998). The limit state equation for the grass revetment is based on the method of Seijffert &Verheij (1998) and uses the thickness of the grass revetment (d_g [m]) and the underlying clay layer (d_k [m]), as well as the quality of both (expressed by coefficients c_E [ms] and c_{RK} [ms]) as input. Also the Manning coefficient (n [sm$^{-1/3}$]) of the inner side material and the friction coefficient (f [–]) of the outer side are

to be estimated. In a second phase the methodology was extended to also incorporate other types of revet ments, such as open stone asphalt, concrete layers, and gabions.

Erosion of the inner slope can occur due to over flow and/or wave overtopping. Overflow is calculated using the weir equation and wave overtopping using the TAW methodology (TAW 2002). Likewise the ero sion mechanism on the outer slope, the strength and the limit state equation depend on the material on the slope. For grass again the method from Seijffert &

Verheij (1998) is used, while formulas for most other materials are based on Pilarczyk (1998).

2.4 Global failure

Each failure mechanism can be described as a function of a number of 'resistance' parameters which can vary according to associated probability distributions, resulting in a statistical variation of the resistance S of the dike section for each failure mechanism. If the probability distribution of each parameter is known, also the probability of a certain combination of parameters is known. Also the probability of the load R – through the synthetic events (see sections 3.1.3 and 3.2.3 for examples) – is known. By combining a resistance S with a load R, the probability of failure can be calculated – if the limit state equation is exceeded – based on the probability of both R and S. When doing this for all possible combinations of R and S and for each failure mechanism, the whole range of probabilities is covered and the global failure probability of the considered dike section is calculated.

The above will now be explained in a more formal way. Denote the probability of a hydraulic boundary condition by p_{Rj}, with j between 1 and the number of synthetic load events (m), and denote the probability of a set of geotechnical parameter values associated with failure mechanism k by p Ski , with i between 1 and

the number of geotechnical parameter sets of failure

mechanism k (n k), with k between 1 and the number

of failure mechanisms (7 in this study).

The size of n k depends on (i) the number of strat

ified samples (5 in our study) and (ii) the number of

parameters l k used to describe failure mechanism k:

The number of parameter sets hence is different for

each failure mechanism k.

For each hydraulic load condition j and for each

failure mechanism k the limit state equation associated

with mechanism k is verified. The resulting failure

probability p jk is the sum of the probabilities of those

geotechnical parameter sets that lead to dike failure:

The failure probability of the boundary condition j

for all failure mechanisms p j is then calculated as: In this equation 7 is the number of failure mechanisms considered here (global and local instability on the inner and outer side counting as separate mechanisms).The total failure probability of the dike section, p, is calculated by: The resulting return period T of dike failure can be calculated as: 2.5 BRES software To execute the necessary calculations a software tool called BRES (Dutch for 'breach') was developed as part of project WL_11_28, funded by Flanders Hydraulics Research (FHR). BRES streamlines the many calculations that are required for the Monte Carlo approach. Its input and output consists of simple text files; graphical information is available as well. A graphical user interface enables a more intuitive set-up of the input and output. 3 APPLICATIONS The method elaborated above was applied to different dike segments in Flanders. The following sections describe only a few aspects of these applications in detail. A full account of the investigations is written in IMDC (2014a, b, 2015a) for the

Meuse River, and in IMDC (2015b) for the Scheldt River at Doel. 3.1 Meuse river 3.1.1 Context In this study, commissioned by FHR, the methodology was applied to the stretch of the Meuse River at the border between Belgium and The Netherlands, the so-called Grensmaas (Fig. 4). It should be noted that the considered dikes are located at the border of the winter bed of the river Meuse. Consequently the distance between the river and the dike can be large (up to more than 1 km). The tested locations (six in total) were identified based on an inventory of general data of the dike, such as dimensions, presence of a revetment, and core material (IMDC 2014a). 3.1.2 Available data Topographically measured cross sections every 100 m were available. Data of some sections had to be completed with the digital elevation model (DEM) of Flanders created by the Flanders Geographical Information Agency (AGIV in Dutch) (AGIV 2004), because of the width of the winter bed. Data from cone penetration tests (CPTs) at regular spatial intervals of the dike were available, as well as borehole data. Rijkswaterstaat

Figure 4. Overview of the study area (Meuse River in

Flanders). Image source: Google, TerraMetrics.

(the Dutch waterways authority) provided the dis

charge time series from the station of Borgharen, just

upstream of the study area. Wind data from the Royal

Dutch Meteorological Institute (KNMI) were available

from the station of Beek.

3.1.3 Load

To determine the water level and flow velocity at

the studied locations, Flanders Hydraulics Research

performed simulations with a 2D hydrodynamic

WAQUA model of the Meuse, using SIMONA ver

sion 2011. The WAQUA model was made available by

Rijkswaterstaat.

The model boundary conditions consisted of syn thetic hydrographs, derived from a statistical analysis of the available discharge time series.

The synthetic hydrographs were scaled versions of a unit hydrograph, determined from the 20 most extreme events in Borgharen. This number was decided to be a suitable compromise between using sufficient extreme hydrographs on the one hand, and using hydrographs that are sufficiently extreme on the other hand. The variation in the hydrographs of these events was taken into account by applying a normal variation on the 'mean' hydrograph, for each value of the discharge. Figure 5 shows an example. More information about the derivation of the synthetic hydrographs is described in IMDC (2014b).

In total, 55 different hydrographs were simulated

with the WAQUA model. Since no correlation was Figure 5. Synthetic events for the Meuse River. The depicted hydrographs are the mean hydrograph plus normal variation. Figure 6. Profile of CPT results along the Meuse dike at location MM174. found between the occurrence of extreme discharges and extreme wind speeds, it was sufficient to only simulate the synthetic hydrographs, and combine the resulting water levels and velocities with the 12 wind direction classes and 11 wind speed classes (different for each direction). This added up to a total of 7200 synthetic events for the dike sections (7200 and not 7260 because in the discretization of the multivariate distribution, the combination of the lowest wind and discharge class is not considered), each with its own probability of occurrence. The phreatic line and wave load were calculated with the methods described in section 2.2.2. 3.1.4 Resistance The geotechnical parameters were derived from the CPTs and the lab results of the borehole samples. For some locations, several CPTs were executed close enough, enabling to derive a mean value and variation instead of having to rely on literature values. For instance, Figure 6 shows a profile of CPT results in the vicinity of MM174. The presence of a layer with higher resistance (indicated between the red lines) is clear.The layer can be classified as coarse sand/gravel. For other locations however, values from other locations along the Meuse, or from literature, had to be assumed. Table 1. Failure probability of section MM89, Meuse River. Failure No. of p T mechanism computations 1/year year global instability outer slope 5 3 × 7200 = 9.0E+05 5.8E-04 1.72E+03 global instability inner slope 5 3 × 7200 = 9.0E+05 0 ∞ piping 5 8 × 7200 = 2.8E+09 0 ∞ erosion outer slope 5 4 × 7200 = 4.5E+06 3.4E-03 2.97E+02 erosion inner slope 5 5 × 7200 = 4.5E+06 8.6E-04 1.16E+03 global failure 4.8E-0.3 2.08E+02

The models used to calculate the limit state are

described in section 2.3.2. For the Meuse case, the

failure mechanism of local instability was not consid

ered because the dike sections are homogeneous, i.e.

there is no top layer that can be pushed of by excessive

phreatic pressure.

3.1.5 Results

Table 1 shows the results of the BRES calculations for

one of the studied sections (MM89).

The insensitivity to piping could be expected since

the underground along the Meuse dike is characterised

by a layer of coarse sand/gravel, which is beneficial for

its resistance to piping.

The relatively high probability of failure for erosion

at the outer slope is due to the fact that there is no

variation for wind taken into account. Since the events

last for almost 4 days, the assumption of a constant extreme wind speed – and correspondingly extreme waves – is likely to be conservative.

Eventually, it can be seen that the inner slope is sta ble, a result found in the other sections as well. The dikes along the Meuse winter bed are relatively low on the inner side. Moreover the dike core consists of a material described as ‘sand-loam’, a mixture of sand and loam, which has both a low hydraulic conductiv ity and high resistance characteristics (ϕ and c), two elements resulting in a very stable slope.

3.2 Scheldt river at Doel

3.2.1 Context

The Maritime Access Division of the Flemish Govern ment is preparing the construction of a new area with controlled reduced tide (CRT) at the left bank of the Scheldt River, the so-called CRT area ‘Doelpolder’. The newly designed ‘ring dike’ of the area is close to the nuclear power plant (NPP) of Doel (15 km northwest from Antwerp). Figure 7 shows the study area.

The ring dike was submitted to the same type of failure analysis as the already existing dikes around the NPP, which were analysed in the framework of the period safety review, the so-called ‘stress test’ (Elec trabel 2011, FANC-BelV 2011). The study also took into account a sequential failure of the Scheldt dike

and the ring dike. Figure 7. Planned layout for the CRT area Doelpolder. The Scheldt dike is currently existing, the to-be-constructed ‘ring dike’ will surround the CRT area. The results will be used in the new stress test of the plant, and will serve as input for the design of the surrounding water evacuation infrastructure. 3.2.2 Available data A local DEM of the projected (designed) situation was available. Outside the project area, the DEM of Flanders (AGIV 2004) was used (Fig.7). Data from CPTs along the alignment of the future ring dike were available, as well as some borehole data, approx. 1–1.5 km southwest of the area.The structure of the ring dike was derived from available design plans. Wind data were taken from the measurements inVlissingen, performed by the KNMI. 3.2.3 Load To determine the hydraulic load on the Scheldt dike, synthetic events (see further below) were simulated with the existing Sea Scheldt model (IMDC et al. 2003a, b, IMDC 2010). This model is implemented in the Mike11 software (DHI, Denmark). For the ring dike, a Mike11 model, coupled to this Scheldt model, was made available by Flanders Hydraulics Research (Coen et al. 2013). The boundary conditions for the hydrodynamic model consisted of 544 synthetic storm events, derived as explained in IMDC (2007) and Blanckaert et al. (2015). Basically, each event was characterized by a water level class, a wind speed class, and a wind direction class. For the Sea Scheldt, there is a strong

correlation between the downstream (extreme) water

level and the wind speed and direction. A perfect

correlation was assumed, so each of the 34 chosen

water level classes corresponded to one wind speed

value. The wind direction was taken separately into

account, divided into 16 classes, resulting into a total of

544 synthetic events. The probability of occurrence of

each event was derived from the directional frequency

curves of the water levels. As an example, Figure 8

shows the downstream boundary water levels of the model for western winds.

As in the Meuse case, the wave load on the Scheldt dike was calculated using Bretschneider's formula (TAW 1985). For the ring dike results from a SWAN model (TUDelft 2010), set up by FHR (Coen et al. 2013), could be used.

3.2.4 Resistance

The angle of repose ϕ was derived from the CPT data. Also, because more than one CPT was performed, the distribution of ϕ could be estimated as well, instead of having to rely on literature. The resulting values were: $\phi = 27.9^\circ$ ($\sigma = 1.67^\circ$), $c = 0$ kN/m^2 ($\sigma = 0^\circ$). Of course, for the to-be-constructed ring dike no data were available. Values were taken from the design calculations: $\phi = 30.8^\circ$ ($\sigma = 3.4^\circ$), $c = 0$ kN/m^2 (no variation). Other geotechnical parameters, such as par ticle size, specific weight or hydraulic conductivity, were derived from laboratory analyses of the borehole samples, performed in the framework of the project, or using 'typical values' from literature if necessary.

Figure 8. Downstream boundary water levels for synthetic storm events, in this case corresponding to western winds.

Table 2. Failure probability of the considered section of the Scheldt dike. Failure No. of p T mechanism computations 1/year year global instability outer slope 5 3 × 544 = 6.8E+04 2.29E-03 4.37E+02 global instability inner slope 5 3 × 544 = 6.8E+04 0 8 local instability outer slope 5 5 × 544 = 1.7E+06 9.80E-06 1.02E+05 local instability inner slope 5 5 × 544 = 1.7E+06 6.76E-10 1.48E+09 piping 5 6 ×

544 = 8.5E+06 0 8 erosion outer slope 5 4 × 544 = 3.4E+05 0 8 erosion inner slope 5 5 × 544 = 3.4E+05 0 8 global failure 2.30E–03 4.34E+02 The models used to calculate the limit state are described in Section 2.3.2. 3.2.5 Results Table 2 shows the most important results of the BRES calculations for the Scheldt dike. It can be argued whether it was necessary to perform the calculations for the failure mechanisms showing zero probability, because from beforehand it was known that the resistance was many times higher than the load. However because of the proximity of the NPP Doel it was requested to explicitly consider all failure mechanisms. Also, the global failure might appear low. However, a sequential failure analysis on the ring dike segments ‘EAST’, ‘SOUTH’and ‘WEST’was performed as well. This resulted in a return period of failure of respectively 4.84E+05, 5.43E+02 and 3.45E+04 year. A consequence analysis was performed if a breach would develop in the southern section, using a 2D surface runoff model (IMDC 2015b). From that analysis, it appeared that the key parts of the NPP site are not compromised, because the site’s platform level is at 8 m TAW and higher. Non-crucial parts, such as the visitors parking, do flood during extreme events. 4 CONCLUSIONS AND RECOMMENDATIONS A probabilistic method was presented to determine the probability of dike failure. This method considers seven different failure mechanisms, and uses multivariate analyses and stratified sampling as statistical instruments. Moreover, it uses several physicallybased numerical models (or response curves derived from calculations with these models) for the failure evaluation and the transformation of boundary conditions to hydraulic loads at the location of the dike section. The methodology was successfully applied at different dike segments in Flanders, both on tidal and non-tidal rivers. The probabilistic approach calculates the global probability of failure of the dike section, but also the probability related to the different failure mechanisms

separately. In addition, the full spectrum of load and

resistance is considered, instead of only the most con

servative ones. These elements provide a clearer view

on those parts of the dike that require additional focus

for possible reinforcement.

The methodology considers many phenomena in detail. Still, further improvement is possible. At the load side ship-induced waves and currents as well as a varying wind profile could be incorporated. At the resistance side, residual strength could be considered (at this moment the dike is already considered as ‘fail ing’at initiation of the failure mechanisms), additional dike revetment materials could be included, or the phe nomenon of water infiltration into the dike could be considered.

AGIV. 2004. Digitaal Hoogtemodel Vlaanderen.

Bishop, A.W. 1955. The use of the slip circle in the stability analysis of earth slopes. Géotechnique, 5(1): 7–17.

Blanckaert, J., Leysen, G., Franken, T., Gullentops, C., Fang, Z., Pereira, F. & Swings, J. 2015. Generation of sets of synthetic events by multivariate stratified sampling. Proc. 36th IAHR World Congress, 28 June–3 July 2015. The Hague, The Netherlands.

Coen, L., Suzuki, T., Altomare, C., Plancke, Y., Peeters, P., Taverniers, E., Verwaest, T. & Mostaert, F. 2013. Advies dijkhoogtes Doelpolder: Deelrapport 1 – Aan vullende scenarioberekeningen met betrekking tot de veiligheid tegen overstromingen in het kader van de aanleg van het GGG Doelpolder. Version 2_0. WL Rap porten, 12_130 (in Dutch). Flanders Hydraulics Research, Antwerp, Belgium.

Electrabel. 2011. Kerncentrale Doel: Rapport weerstand stesten – Bijkomende veiligheidsherziening, 31 October 2011 (in Dutch).

FANC-Bel V. 2011. Belgian Stress tests: National Report for Nuclear Power Plants. Federal Agency for Nuclear Control.

Geo-Slope. 2012. SEEP/W. Groundwater Seepage Analysis.

IMDC. 2007. Onderzoek naar maximale waterstanden van de Schelde te Doel. Study for Tractebel Engineering. I/RA/14098/06.152/JBL (in Dutch).

IMDC. 2010. PSR Doel 2010 Flood Hazard: Technisch Rapport. Study for Tractebel Engineering. I/RA/14157/10.198/JBL (in Dutch).

IMDC, Tractebel Engineering & Jan Maertens bvba. 2010. Onderzoek naar de bresgevoeligheid van de Vlaamse winterdijken – Deelopdracht 5: Opstellen van een wetenschappelijk verantwoorde en praktisch haalbare methode. For Flanders Hydraulics Research.

I/RA/11279/08.009/RVL (in Dutch). IMDC. 2014a. Het berekenen van de faalkansen van de winterdijken langs de Maas – Deelopdracht 1: Inventarisatie Maas. For Flanders Hydraulics Research. I/RA/11400/13.111/MOE (in Dutch). IMDC. 2014b. Het berekenen van de faalkansen van de winterdijken langs de Maas – Deelopdracht 2: Verbetering probabilistische methode. For Flanders Hydraulics Research. I/RA/11400/13.181/TGO (in Dutch). IMDC. 2015a. Het berekenen van de faalkansen van de winterdijken langs de Maas – Deelopdracht 3: Probabilistische faalkansberekening van de winterdijken langs de Maas. For Flanders Hydraulics Research. I/RA/11400/13.301/TGO (in Ducth). IMDC. 2015b.

Stresstest Kerncentrale Doel i.h.k.v. GGG: Gevolgstudie bij dijkfalen. Study for Maritime Access Division. I/RA/14209/14.217/TGO (in Dutch). Kortenhaus, A., Weissmann, R., Oumerai, H. & Rihwein, W. s.d. Probabilistische Bemessungsmethode für seedeiche: Prodeich (in German). Pilarczyk, K.W. 1998. Dikes and Revetments. Rotterdam: Balkema. Seijffert, J.W. &Verheij, H. 1998. Grass covers and reinforced measurements. In Krystian W. Pilarczyk (ed.), Dikes and Revetments: 289–302. Rotterdam: Balkema. Sellmeyer, J.B. 1988. On the mechanism of piping under impervious structures, PhD thesis Delft. TAW. 1985. Leidraad voor het ontwerpen van rivierdijken: Deel 1, Bovenrivierengebied. Staatsuitgeverij, 'sGravenhage, ISBN: 90-12-05169-X (in Dutch). TAW. 1998. Technisch rapport erosiebestendigheid van grasland (in Dutch). TAW. 2001. Technisch rapport TR-19: Technisch Rapport Waterkerende Grondconstructies (in Dutch). TAW. 2002. Technisch rapport golfoploop en golfoverslag bij dijken (in Dutch). TAW. 2004. Technisch rapport waterspanningen bij dijken (in Dutch). THV Sigma. 2003b. Actualisatie van het Sigmaplan – Deelopdracht 3: Hydrologische en hydraulische modellen, Volume 2a: Hydraulica Scheldebekken. Study for the Sea Scheldt division of the Ministry of the Flemish Community. I/RA/11199/03.003/SME (in Dutch). THV Sigma. 2003c. Actualisatie van het Sigmaplan – Deelopdracht 3: Hydrologische en hydraulische modellen, Volume 2b: Hydrodynamisch model Rupelbekken. Study for the Sea Scheldt division of the Ministry of the Flemish Community. I/RA/11200/03.002/SME (in Dutch). TNO. 2003a. Theoriehandleiding PC-Ring. Versie 4.0. Deel A: Mechanismenbeschrijving. (in Dutch). TNO. 2003b. Theoriehandleiding PC-Ring. Versie 4.0. Deel B: Statistische modellen. (in Dutch). TUDelft. 2010. SWAN (Simulating WAves Nearshore): a third generation wave model. Copyright © Delft University of Technology. Available free of charge at http://www.swan.tudelft.nl. Verruijt A. 1995. Computational Geomechanics. Dordrecht: Kluwer Academic Publishers. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Developing a 100-point scoring system for quality assessment of ADCP

streamflow measurements

H. Huang

Teledyne RD Instruments, Poway, CA, USA

ABSTRACT: This paper presents a 100-point scoring system for quality assessment of moving-boat ADCP streamflow measurements.The proposed scoring system utilizes results from descriptive statistics and uncertainty analysis on a measurement; it employs a newly developed median-unbiased uncertainty estimator as a major quality indicator; it also considers four other quality indicators.The rating scale in the proposed 100-point scoring system is designed to be an analogy to the well-known 100-point wine rating scale. The score point calculators for the five quality indicators are presented. The score for a measurement is the sum of all points assigned to the five quality indicators. Results for moving-boat ADCP streamflow measurements on five rivers are presented as application examples. The proposed scoring system provides a quantitative measure of the overall quality of moving-boat ADCP streamflow measurements.

1 INTRODUCTION

The use of acoustic Doppler current profilers (often known as ADCPs) for streamflow measurements has increased significantly in the past ten years. The need to establish a framework with which to assess the quality of ADCP streamflow measurements becomes important. A quality assessment framework may con sist of three components: (1) descriptive statistics, (2) uncertainty analysis, and (3) a scoring system. Descriptive statistics describes the basic features of the data collected in a measurement, which provides sum maries and graphics analysis and forms the basis of quantitative analysis of the measurement. The uncer tainty analysis may be considered as an inferential statistics. This is because the result from uncertainty analyses is often used to infer the error limit or con fidence interval associated with the measured value.

A scoring system assigns points to a number of qual ity indicators; the sum of all points is the score for a measurement. A scoring system provides a quantita tive measure of the overall quality of a measurement. This paper considers scoring systems only. However, a scoring system may utilize results from descriptive statistics and uncertainty analysis.

Randall (2013) developed a scoring system which becomes a quality evaluation tool recommended by the Australia Water Information Standards Busi ness Forum (WISBF 2013). A similar scoring sys tem was developed and implemented by the New Zealand National Environmental Monitoring Stan dards (NEMS 2013). The WISBF scoring system evaluates eight quality indicators, each of which may be assigned a point from 1 to 5 based on a prede fined rule. The score for a measurement is the sum of all points assigned to the eight quality indicators. A measurement will be graded as ‘good’ if its score is between 8–10 points, ‘fair’ if 11–19 points, ‘poor’ if 19–34 points, and ‘unrated’ if 35+ points. The NEMS scoring system evaluates twelve quality indicators, each of which may be assigned a point from 0 to 3 based on a predefined rule. The score for a measurement is the sum of all points assigned to the twelve quality indicators. A measurement will be graded as ‘good’ if its score is <4 points, ‘fair’ if between 4–12 points, and ‘poor’ if >12 points. Although either of the scoring systems has its own merits, three observations can be made. First, both scoring systems do not use the discharge measurement uncertainty as a quality indicator. According to the International Organization for Standardization (ISO) document: “Guide to the Expression of Uncertainty in Measurement (GUM, JCGM 2008),” uncertainty is the most important quality indicator for measurements. The WISBF scoring system includes the coefficient of variation of discharge (COV of discharge Q), which measures the dispersion of the transect discharges. However, the COV of Q is not the same as the uncertainty of the measured discharge. Second, in both scoring systems, a low score is associated with a high quality and a high score is associated with a low quality. This quality scaling is not easy to understand, particularly for ADCP users in some countries where a low score is often associated with a low quality and a high score is often associated with a high quality.Third, both scoring systems include some quality indicators that rely on operator’s visual observation or subjective judgments. One of such indicators is the channel description and the other is the goodness of fit of velocity profile data with a extrapolation model. The visual observation or

subjective judgments of these indicators may cause

inconsistency in scoring among different operators.

This paper presents a 100-point scoring system for

quality assessment of moving-boat ADCP streamflow

measurements. In the following, section 2 describes

the quality indicators used in the proposed scoring

system; section 3 describes the score scale and point

calculators (formula); and section 4 presents five

application examples of the proposed scoring system

with discussions.

This paper does not make any discussions on

ADCP technology and its use in streamflow measurements. Readers may refer to Simpson (2001), Oberg et al. (2005), and Mueller & Wagner (2009) about ADCP technology and moving-boatADCP streamflow measurements.

2 MEASUREMENT QUALITY INDICATORS

There are a number of quantities that may be used as quality indicators for ADCP streamflow measurements. After carefully analyzing a number of candidates, including the indicators used in the WISBF and NEMS scoring systems, the following five statistical quantities are selected as quality indicators: the uncertainty of the measured discharge (or known as the total Q), the percentage of the ADCP directly measured discharge with respect to the total Q (denoted by $\%Q_{measured}$), the percentage of the edge discharge with respect to the total Q (denoted by $\%Q_{edge}$), the percentage of the bad cells with respect to the total cells in the measured region (denoted by %bad-cells), and the root mean square percentage error (denoted by RMSPE) of the normalized unit discharge profile data with respect to the normalized 1/6 power law. The second and third indicators are straightforward and can be obtained from the ADCP software (e.g. WinRiver II, TRDI 2012). The fourth indicator is the sum of all bad cells in the measured region, including the cells in bad or lost ensembles. The sub-sections below address the uncertainty and RMSPE in detail.

2.1 Uncertainty of the measured discharge

Under a steady flow condition, a moving-boat ADCP streamflow measurement usually involves multiple transects, producing the transect discharges $Q_1, \ldots, Q_n$ (where n is the number of transects, or the sample size). The mean discharge $\bar{Q}$ is taken as the measured discharge. The uncertainty of the measured discharge needs to be estimated. This is a classical small sample problem: estimating the uncertainty associated with the sample mean when the sample size is small and the population standard deviation is unknown. The uncertainty is traditionally calculated as the t-based uncertainty (e.g. GUM, JCGM 2008). However, Huang (2010) revealed that the use of the t-based uncertainty caused a paradox in measurement uncertainty analysis, i.e. the uncertainty evaluation based on the t-based uncertainty is not compatible with that based on the z-based uncertainty, particularly for small samples. The paradox can be resolved by using a mean-unbiased estimator or a median-unbiased estimator of the z-based uncertainty (Huang 2010, 2014, 2015a). The median-unbiased estimator is considered as the optimal estimator for uncertainty-based measurement quality control (Huang 2015a); it is recently recommended for the Type A uncertainty estimation of moving-boat ADCP streamflow measurements (Huang, in press). Therefore, the median-unbiased estimator is employed in the proposed scoring system. The median-unbiased estimator, denoted by U , is calculated as (Huang 2015a) where s = sample standard deviation of transect discharges, $s = \sqrt{\Sigma (Q_i - Q)^2/(n - 1)}$; and C_{OE} = estimator coefficient (Huang 2015a) The relative uncertainty, denoted by RU, is the ratio between U and Q Because it is calculated using the discharge data collected at a site, RU is known as the Type A uncertainty according to GUM (JCGM 2008). RU accounts for all sources of random errors in the discharge measurement at a site (Huang, in press). It should be mentioned that, traditionally, the relative maximum residual (RMR) of transect discharges is used as a quality indicator and the four transects RMR ≤ 5% is used as a quality control criterion for moving-boatADCP streamflow measurements (e.g. Lipscomb 1995, USGS 2002, Oberg et al. 2005, Mueller & Wagner 2009). A recent study (Huang 2015b) demonstrated that the two sample statistics RMR and RU are highly correlated. Huang (2015b) presented two alternative quality control criteria: an uncertainty-based and a residual-based, and suggested using either one of them, but not the both. The maximum permissible uncertainty in the uncertainty-based criterion is a constant and can be met with increasing the number of transects, which is intuitive for quality control. In contrast, the maximum permissible residual in the residual-based criterion is a function of the number of transects; it is increasing with increasing the number of transects, which is unintuitive for quality control. Therefore, the proposed scoring system does not employ RMR as a quality indicator.

2.2 RMSPE of the normalized unit discharge profile

An ADCP discharge calculation algorithm utilizes velocity extrapolation methods to estimate the dis charge in the unmeasured surface or bottom layers. For a wide, fully developed open channel flow, the velocity profile is best described by the 1/6 power law (Chen 1991). Therefore, the 1/6 power law has been used as a default extrapolation model in ADCP discharge calculation algorithms (e.g. TRDI 2012). However, a flow at a site may not exactly follow the 1/6 power law. Muller (2013) recently developed a methodology to combine all ensemble velocity profiles on a single plot, known as the normalized unit discharge (UQ) plot. The methodology and the UQ plot have been implemented in the extrap software (Muller 2013) and the Q-View software (TRDI 2015). The UQ plot helps examine the goodness of fit of velocity profile data with the 1/6 power law. In principle, the better the fit, the more accurate the calculated discharge. The good ness of fit can be quantitatively measured by the root mean square percentage error (RMSPE) of the nor malized UQ profile data with respect to the 1/6 power law

where m = number of cell groups (20 in the Q-View software, TRDI 2015) for a normalized UQ plot;

i = index for cell groups; $\tilde{q}_i$ = normalized cell dis charge q_i ; and $q_{p,i}$ = 1/6 power law for the normalized UQ. $\tilde{q}_i$ is calculated using ADCP cell and ensemble discharge data

where q = depth-averaged cell discharge at an ensem ble; q is the ensemble discharge; and H is the water depth at the ensemble.

The cell discharge q_i is calculated as the cross product of the water (cell) velocity and boat velocity vectors (e.g. TRDI 2012). It is assumed that the cell cross-product also follows a power law as the cell

velocity does (RDI 1992). Thus, if the cell velocity is

assumed to follow the 1/6 power law, the cell discharge

q(z) will also follow the 1/6 power law

where a = coefficient; and z is the upwards coordinate

with zero at the river bottom. The depth-averaged cell

discharge q can be calculated as

Thus, the 1/6 power law for the normalized UQ is Table 1. Analogy between the 100-point wine rating scale and the proposed 100-point rating scale forADCP streamflow measurements. ADCP streamflow Score Wine (Newberry 2014) measurements 96–100 Classic; a great wine; A + : excellent extraordinary 90–95 Outstanding; a wine of A: very good superior character and style 80–89 Very good; a wine with B: good special qualities 70–79 Mediocre; average; C: fair harmless 60–69 Below average; D: meet minimum deficient criterion 50–59 Just downright F: fail to meet (or unacceptable minimum criterion; below 59) unacceptable Note that z i /H is the normalized depth ranging from 0 (the river bottom) to 1 (the water surface). 3 THE 100-POINT RATING SCALE The rating scale in the proposed 100-point scoring system is designed to be an analogy to the well-known 100-point wine rating scale, which was originally developed by Robert Parker, one of the most influential wine critics in the world, after the American high school grading system, assuming people would easily understand that a rating of 95 or above translates to an A+, etc. (Newberry 2014). Table 1 shows the analogy between the two 100-point rating scales. Among the five quality indicators considered in the proposed 100-point scoring system, the uncertainty is the most important measure of the measurement quality; it is recommended for statistical quality control, i.e. making the decision on acceptance or rejection of the measured discharge (Huang 2015b). Therefore, the uncertainty is given the greatest weight in the 100point rating scale with a point range 0–80. Each of the other four quality indicators is given a point range 0–5. Table 2 summaries the typical ranges of the five quality indicators and their point ranges. Equations (9) to (12) show the point calculators (formulas). Most of the formulas are simple linear relations except the one for the uncertainty. Figure 1 shows the calculated point associated with the uncertainty as a function of RU.

Table 2. Summary of the typical ranges of the quality indicators and point ranges.

Quality Typical Point Point

indicator range (%) range calculator

RU 0.5–25 0–80 Eq. (9)

%Q measured 50–85 0–5 Eq. (10)

%Q edge 0–20 0–5 Eq. (11)

%bad-cells 0–20 0–5 Eq. (11)

RMSPE 0–10 0–5 Eq. (12)

Figure 1. Plot of point calculator for RU.

where x stands for a quality indicator value in percentage.

The score for a measurement is the sum of all points calculated by Eqs. (9) to (12), which will be reported and used to grade the measurement according to the proposed 100-point rating scale in Table 1. However, an exception is that, if RU > 4.09%, but the sum of all five indicators' points is equal to or greater than 60 point, assign 59 point as the score. This is because, 4.09% is the maximum permissible RU, derived from the existing four transects RMR ≤ 5% criterion (Huang 2015b); RU ≤ 4.09% is assumed to be the quality con trol criterion. That is, if RU > 4.09%, the measured discharge should not be accepted or the measurement would be rated as F (fail to meet the minimum crite rion, which is unacceptable; see Table 1). In practice, an operator must make sure that the quality control criterion be met (i.e. RU ≤ 4.09%) before leaving the site. Assume that an operator initially made four tran sects and found that the four transects' RU > 4.09%. The operator should make additional transects until the quality control criterion is met.This procedure ensures that the estimated score will be greater than 60 points. 4 APPLICATION EXAMPLES The proposed 100-point scoring system was tested with the moving-boat ADCP streamflow measurements on five river sites. Table 3 shows the site and measurement information of the five rivers. Notice that the sites range from a small channel (the California irrigation canal) to large rivers such as Mississippi River andYangtze River. Tables 4 to 8 show the scores and grades as a function of the number of transects in a measurement. A measurement may consist of all transects made, the first 4 transects, or the first 2 transects. All ADCP velocity data were referenced to the bottom tracking. Three observations can be made for these five test examples. First, as expected, the uncertainty dominates the score or grade. The lower the uncertainty, the higher the score or the grade is. Second, in general, the uncertainty decreases with increasing the number of transects in a measurement. Accordingly, the score increases with increasing the number of transects. However, because of the randomness of transect discharges, there might a case where a measurement with less transects (say two transects) may have a lower uncertainty and higher score than a measurement with more transects (say four transects) does. The measurement on the California irrigation canal is such a case (Table 4) where the score associated with the four transects is 78.3, graded as C, whereas the score associated with the two transects is 88.8, graded as B. Third, also as expected, the four other quality indicators have much less contribution to the score than the uncertainty does. In addition, the values for these indicators are not sensitive to the number of transects in a measurement. This suggests that, for a given ADCP used at a site with the same user's settings, the influence of these four indicators on the measurement quality is approximately fixed for the site.

The application examples on the five rivers demonstrated the feasibility of the proposed scoring system. However, the datasets tested only cover a limited range of measurement conditions or data qualities.Tests with a large number of datasets that covers a large range of measurement conditions or data qualities are needed to further evaluate and gain confidence with the proposed scoring system. 5 CONCLUSIONS The proposed 100-point scoring system provides a quantitative measure of the overall quality of movingboat ADCP streamflow measurements. It is easy to understand and easy to use. It can be implemented in a computer program or an Excel spreadsheet. The application examples on the five rivers demonstrated the feasibility of the proposed scoring system. An advantage of the proposed scoring system is that the score or grade for a given measurement is determined by the system without user’s subjective judgments. That is, the system provides the same

Table 3. Site and measurement information of five rivers.

River	Test date	ADCP model	Total transects	Discharge ($m^3 s^{-1}$)	Width (m)	Maximum depth (m)
California irrigation canal	February 4, 2004	2000 kHz StreamPro	4	1.18	6.8	1.4
Han River, Korea	May 30, 2013	1200 kHz Rio Grande	4	321.7	518	4.0
Manilla River, Australia	December 4, 2013	600 kHz RiverRay	8	47.3	55	1.8
Mississippi River, USA	October 15, 2009	600 kHz Rio Grande	11	14639	690	37
Yangtze River, China	September 2, 2002	300 kHz Monitor	6	11472	490	55

Table 4. Scores and grades for measurements on the California irrigation canal.

Measurement	RU (%) value (point)	%Q measured value (point)	%Q edge value (point)	%bad cells value (point)	RMSPE (%) value (point)	Score (total points)	Grade
All 4 transects	2.61 (62.4)	69.71 (2.8)	2.54 (4.4)	1.0 (4.8)	2.0 (4.0)	78.3	C, fair
First 2 transects	1.37 (72.7)	70.89 (3.0)	2.59 (4.4)	4.0			

(4.8) 2.0 (4.0) 88.8 B, good

Table 5. Scores and grades for measurements on the Han River. RU (%) %Q measured %Q edge %bad cells RMSPE (%) Score

Measurement value (point) value (point) value (point) value (point) value (point) (total points) Grade

All 4 transects 0.99 (75.9) 75.91 (3.7) 0.05 (5.0) 4.0 (4.0) 2.0 (4.0) 92.6 A, very good

First 2 transects 3.02 (58.9) 76.16 (3.7) 0.05 (5.0) 4.0 (4.0) 2.0 (4.0) 75.7 C, fair

Table 6. Scores and grades for measurements on the Manilla River. RU (%) %Q measured %Q edge %bad cells RMSPE (%) Score

Measurement value (point) value (point) value (point) value (point) value (point) (total points) Grade

All 8 transects 1.78 (69.3) 62.99 (1.9) 1.55 (4.6) 0 (5.0) 4.8 (2.6) 83.4 B, good

First 4 transects 2.13 (66.4) 62.84 (1.8) 2.11 (4.5) 0 (5.0) 4.8 (2.6) 80.3 B, good

First 2 transects 6.54 (36.1) 62.92 (1.8) 2.6 (4.3) 0 (5.0) 4.8 (2.6) 49.9 F, fail

Table 7. Scores and grades for measurements on the Mississippi River. RU (%) %Q measured %Q edge %bad cells RMSPE (%) Score

Measurement value (point) value (point) value (point) value (point) value (point) (total points) Grade

All 11 transects 0.48 (80) 85.24 (5.0) 0.06 (5.0) 3.0 (4.3) 4.06 (3.0) 97.2 A + , excellent

First 4 transects 0.96 (76.2) 88.10 (5.0) 0.08 (5.0) 3.0 (4.3) 4.06 (3.0) 93.4 A, very good

First 2 transects 0.57 (79.4) 87.93 (5.0) 0.05 (5.0) 3.0 (4.3) 4.06 (3.0) 96.7 A + , excellent

Table 8. Scores and grades for measurements on the Yangtze River. RU (%) %Q measured %Q edge %bad cells RMSPE (%) Score

Measurement value (point) value (point) value (point) value (point) value (point) (total points) Grade

All 6 transects 0.95 (76.2) 80.10 (4.3) 0.64 (4.8) 2.0 (4.5) 1.0 (4.5) 94.4 A, very good

First 4 transects 1.56 (71.1) 80.48 (4.4) 0.57 (4.9) 2.0 (4.5) 1.0 (4.5) 89.4 B, good

First 2 transects 4.84 (45.0) 81.23 (4.5) 0.51 (4.9) 2.0 (4.5) 1.0 (4.5) 59 (63.4)* F, fail

* assigned the score as 59 because the estimated RU 4.84% does not meet the quality control criterion 4.09%, although the

calculated total points is 63.4.

assessment on the overall quality of a measurement

regardless of users.

The proposed scoring system should not be used

alone when assessing a measurement. It should be used with descriptive statistics and uncertainty analysis which provide insights on the quality of the data collected at a site.Tests with a large number of datasets that covers a large range of measurement conditions or

data qualities are needed to further evaluate and gain

confidence with the proposed scoring system.

ACKNOWLEDGMENTS

The author would like to thank two anonymous review

ers for their helpful comments. The field test at the

California irrigation canal site was organized by the

California Polytechnic State University; the field test

at the Han River site was organized by the Hydro

logical Survey Center (HSC), Korea; the field test

at the Manilla River site was organized by the NSW

Office of Water, Australia; the field test at the Yangtze River site was organized by the Yangtze (Changjing) Water Resources Commission, China. The author was engaged in the field tests at these four sites. The data from the field test at the Mississippi River site was provided by Jeffrey Den Herder of Teledyne RD Instruments.

Chen, C.L. 1991. Unified theory on power laws for flow resistance. Journal of Hydraulic Engineering 117(30): 371–389.

Huang, H. 2010.A paradox in measurement uncertainty anal ysis. In: Global Measurement: Economy & Technology 1970–2010 Proceedings (DVD). The 2010 Measurement Science Conference, Pasadena, CA, March 25–26, 2010.

Huang, H. 2014. Uncertainty-based measurement qual ity control. Accreditation and Quality Assurance 19(2): 65–73.

Huang, H. 2015a. Optimal estimator for uncertainty-based measurement quality control. Accreditation and Quality Assurance 20(2): 97–106.

Huang, H. 2015b. Statistical quality control of streamflow measurements with moving-boat acoustic Doppler current profilers. Journal of Hydraulic Research 53(6): 820–827.

Huang, H. in press. Estimation of Type A uncertainty of moving-boat ADCP streamflow measurements, River

Flow 2016: proceedings of the 8th International Confer

ence on Fluvial Hydraulics. St. Louis, MO, July 12–15,

2016.

Joint Committee for Guides in Metrology (JCGM) 2008.

Evaluation of measurement data – Guide to the expression

of uncertainty in measurement (GUM 1995 with minor

corrections). Geneva, Switzerland. Lipscomb, S.W. 1995. Quality assurance plan for discharge measurements using broadband acoustic Doppler current profilers. Open-File Report, 95-701, U. S. Geological Survey (USGS), Reston, VA. Mueller, D.S. & Wagner, C.R. 2009. Measuring discharge with acoustic Doppler current profilers from a moving boat. U.S. Geological Survey Techniques and Methods 3–A22, 72 p. Mueller, D.S. 2013. extrap: Software to assist the selection of extrapolation methods for moving-boatADCP streamflow measurements. Computers & Geosciences, 54, 211–218. National Environmental Monitoring Standards (NEMS) 2013. Open Channel Flow Measurement: Measurement, Processing and Archiving of Open Channel Flow Data, Version 1.1, June 2013, New Zealand. Newberry, B. 2014. What you need to know about the 100-point wine system. The Savory, available from

An alternate approach for assessing impacts of climate change on water resources: Combining hazard likelihood and catchment sensitivity

B. Grelier & G. Drogue

LOTERR, Université de Lorraine, Metz, France

M. Pirotton, P. Archambeau & B. Dewals

Hydraulic in Environmental and Civil Engineering (HECE), University of Liège, Liège, Belgium

ABSTRACT: In this paper we present preliminary results of an alternate approach to the conventional top-down

assessment of climate change impact on water resources driven by climate models. A robust transfer function

linking atmospheric circulation indices and surface climate is used to predict mean areal monthly precipitation

and air temperature estimates at a multi-centennial time-scale (1659–2100), including 13 CMIP5 climate models

and two RCP scenarios. By integrating “paleoclimate” reconstructions in the current knowledge of likelihood of

climate change, climate hazard assessment is treated as a large number of plausible climate changes instead of

being solely expressed as individual CMIP5 projections. Catchment sensitivity to climate change is regarded as

changes in peak flows and low flows in a sub-catchment of the transnational Meuse catchment. By combining

likelihood of climate change with the knowledge of the sensitivity of a given catchment it is possible to develop

a more robust decision-making in water management.

1 INTRODUCTION

IPCC climate projections have been developed for two

decades in order to evaluate the possible evolution

of the climate on the basis of socio-economic sce

narios. Taking into account many uncertainties which

these climatic projections contain, it is advisable to

be careful about their applicability in adaptation deci

sions as the range of projections does not represent

the full range of possible climate changes (Brown &

Wilby, 2012; Prudhomme et al., 2010). Therefore,

IPCC climate projections should not be used like only

source for assessing climate hazard and for providing

useful climate information to stakeholders for water management planning. For instance sampling changes in climate with “paleoclimate” reconstructions could also inform research studies thinking on adaptation to climate change (Smith et al., 2007).

In that perspective, we evaluate the climate haz ard of a mesoscale catchment through a data-driven approach combining paleoclimate reconstructions and climate models run in the framework of the fifth Cou pled Model Intercomparison Project (CMIP5). The originality of our approach resides in creating a contin uum between paleoclimate records and climate model outputs by using the same atmospheric indices to statistically downscale climate surface variables (pre cipitation and air temperature) over a period extending from the Maunder minimum (AD 1659) until the end of the 21st century (2100).This allows sampling paleo

climate variability as well as future climate variability and experiencing the impact of both climate conditions and variability on catchment hydrology. 2 STUDY AREA We demonstrate our approach in the Ourthe river catchment located in southern part of Belgium (Fig. 1). This catchment is part of the transnational Meuse river catchment. Issues of water resources management under climate change are pregnant in this catchment, due to its transnational position and because this catchment hosts a major population (almost 9 million people are living in this catchment) as well as many industry and agricultural activities (Bauwens et al., 2014). The Ourthe catchment is considered at the Tabreux gauging station, controlling a drainage area of 1607 km 2 (Fig. 1). The Ourthe river begins its course in the Ardennes massif, whose maximum elevations reach almost 700 meters above sea level. Mean annual precipitation ranges from 900 mm/year to 1200 mm/year over the Ourthe catchment at Tabreux. Catchment geology is dominated by schist and sandstone formations whose aquifer capacity is weak. Towards the end of the 21st century, global warming scenarios show a decrease in summer discharge, partially because of the diminished buffering effect by the snow pack, whereas high flows are expected to increase due to an enhancement of winter precipitation (Driessen et al., 2010). Nevertheless, those trends are to be taken with caution

Figure 1. Location of the Ourthe river catchment within the

Meuse catchment at Borgharen. Grid points are also shown.

since they are based on only one climate model run

available at a high resolution.

3 DATA

With an aim of considering past and future climate con

ditions and variability in future climate hazard assess

ment, a large array of data sources was considered

(Table 1). Since Kastendeuch (2007) shown a good

agreement between monthly pressure fields and pre

cipitations/air temperatures data over Western Europe,

long instrumental and reconstructed pressure fields

have been collected over a period extended from 1659

to 2005 (Luterbacher et al., 2002; Berrisford et al.,

2009). Note that in the study area correlation coef

ficients between Luterbacher data and atmospheric

pressure data from meteorological stations (Table 1)

are very strong ($r \geq 0.96$). For documenting future cli

mate variability we also collected outputs of 13 climate

models forced with RCP 4.5 and RCP 8.5 radiative

scenarios made available in the framework of CMIP5 experiment. CMIP5 is a multi-model intercompari

son project providing a state-of-the-art context for

studying the response of coupled atmosphere–ocean

general circulation models (AOGCMs) to climate forc

ing, such as increasing in atmospheric greenhouse

gases (Ullmann et al., 2013).

4 SETTING THE EXPERIMENT

4.1 A regression-based statistical downscaling approach

The statistical downscaling methodology carried

out for extending back and ahead knowledge of Table 1. Monthly climatological dataset used in this study. Variables Location Type Atmospheric Maastricht (NL) Instrumental (1906–1999) pressure Leeuwarden (NL) Instrumental (1951–2014) (SLP* and Bron (F) Instrumental (1949–2000) Z500**) Wurzburg (D) Instrumental (1949–2014) Ste-Adresse (F) Instrumental (1967–2007) Grid points Reconstruction and reanalysis (1659–1999) Grid points ERA-Interim (1979–2005) Grid points CMIP5 models (1979–2100) Precipitation Ourthe Gridded data (1961–2013) catchment Air Ourthe Gridded data (1961–2013) temperature catchment * Sea Level Pressure. ** Geopotential height at 500 hPa. Figure 2. Flowchart of the regression-based statistical downscaling model used for extending back and ahead knowledge of monthly catchment precipitation and air temperature. precipitation and air temperature is described in Figure 2. A transfer function linking monthly precipitation and air temperature data to circulation patterns is established according to the predictors found by Kastendeuch (2007) for Western Europe. Mean absolute atmospheric pressure, Y pressure gradient force (PGF) component and X PGF component are used as predictors of atmospheric circulation. They are calculated for the closest grid point (5 ∘ E, 50 ∘ N) to the catchment centroïd (Fig. 1). Atmospheric predictors are calculated from sea level pressure (SLP) data and from geopotential height at 500 hPa (Z500) data. A multiple linear regression (MLR) analysis is carried out for each month on the 1961–2005 period,

Figure 3. Methodology used to transform daily precipita

tion and air temperature time series on the baseline period (1975–2005) according to the delta change and the climate hazard assessment approaches.

using reconstructed pressure data and reanalysis data from Luterbacher et al. (2002) and Berrisford et al. (2009). In order to avoid co-linearity among predic tors, MLRs are fitted in a stepwise mode to retain only predictors that provide a significant level of explained variance. If necessary, prior to estimate precipitation and air temperature in extrapolation mode, climate model atmospheric pressure data are debiased through a quantile-quantile approach (Fig. 3).

Bootstrap series of reanalysis and simulated pres sure data are linked via a linear regression model whose parameters are applied to future pressure time series in order to produce corrected series. Then, the monthly MLRs are provided with raw/corrected climate model pressure data series as well as with historical data series (Fig. 3). For each month, this cal culation protocol provides a set of 347 monthly values for the past (1659–2005) and a set of 2470 monthly values (2 RCP scenarios – 4.5 and 8.5 –× 13 climate models × 95 years) for the future (2006–2100).

4.2 Transformation of the baseline daily precipitation and air temperature time series

This objective is achieved with the delta change method (Fig. 3). This method is a transformation that scales baseline climate time series to obtain series that are representative of mean reconstructed and projected climate conditions. We used additive (multiplicative) anomalies between actual and extended values for air temperature (precipitation). Anomalies are obtained by comparing monthly 30-year averages of precipitation and air temperature calculated from the historical and future datasets to the monthly values averaged over the baseline period. As a result, 2006 precipitation and 2006 air temperature anomalies (i.e. climate change factors) are computed for each month: 316 anomalies applies for the past over the period 1659–1688 to 1975–2004, and 1690 anomalies applies for the future over the period 2006–2035 to 2070–2099 (2 RCP scenarios – 4.5 and 8.5 – × 13 climate models × 65 datasets of 30 years). Computed monthly anomalies of each 30-year dataset are applied to the daily values of the corresponding months over the whole baseline period. By repeating this procedure with the 2006 datasets and related climate change factors, we obtained 2006 transformed precipitation and air temperature time series. 4.3 Hydrological modelling 4.3.1 The rainfall-runoff model In order to estimate the catchment sensitivity to climate hazard we run a lumped and parsimonious rainfallrunoff (RR) model. We used the GR4J model which has been tested on a large sample of French catchments and proves to be a well-performing fixed model structure (Perrin et al., 2003). The GR4J model represents the transformation of the precipitation into streamflow over a catchment using a structure having two stores, a production store and a routing store, respectively acting as a filter that transforms part of the rainfall input into a fast catchment streamflow response and a slow catchment streamflow response. The model has been calibrated by using the Broyden– Fletcher–Goldfarb–Shanno algorithm (hill climbing optimization technique; see Byrd et al., 1995). The parameter optimization is obtained through a multistart procedure and by using the Kling-Gupta Efficiency (KGE) criterion as objective function (Gupta et al., 2009). The arithmetic (logarithmic) version of the KGE is used for optimizing model parameters in high (low) flows. Available discharge data allow optimizing model parameters over the 1988–2005 period (the first year is used for model warm-up). The RR model is provided with mean areal precipitation and mean areal temperature-based potential evapotranspiration (Oudin PET). In agreement with the high KGE values in calibration (more than 0.96) we decided to use the two parameters sets in extrapolation over the 1975–1987 period in order to get two daily discharge series (one representative of high flows and one representative of low flows) for the baseline period.

Figure 4. Example of wet, dry and intermediate 5-year sub-periods selected according to the aridity index for climate-based

rainfall-runoff model optimization. Example of the Ourthe catchment at Tabreux.

4.3.2 A climatologically robust approach for rainfall-runoff model parameterization

As model parameters and extrapolation capacity are depending to climate conditions of the calibration period (Coron et al., 2012), it is necessary to param eterize the model according to climate conditions representative to what could occur under past climate conditions and under future climate conditions. There fore, following the methodology proposed by Brigode et al. (2013), three 5-year sub-periods of the base line period are identified as dry, intermediate and wet according to the aridity index calculated as the ratio between mean annual PET and mean annual precipita tion (Fig. 4). For each sub-period, two parameter sets are optimized according to the two aforementioned version of the KGE criterion. In extrapolation mode, each 30-year moving period is defined as a dry, an intermediate and a wet period in comparison to the mean climate conditions of the baseline period. The assignment of an appropriate climate type to a given 30-year period is made according to the standard devi ation of the aridity index during the 1900–2005 period

(Fig. 7). Therefore, if the residual value of a 30-year mean aridity index is outside the range of one standard deviation of mean, the period is considered as wet or dry. If the residual value is in the range, the period is considered as intermediate.

4.3.3 Calculation of hydrological indices

For high flows, we retained annual maximum peak flows values. For low flows, we computed the annual lowest mean discharge for 7 consecutive days (MAM7). As shown by Figures 5–6, there is a good agreement between observed and simulated values for peak flows and MAM7 discharges during the gaug ing measurement period. For annual maximum peak flows we compute peak flows with 25-year (Q25) and 100-year (Q100) return periods as made by Ernst et al.

(2010) in the framework of their micro-scale flood risk
Figure 5. Observed vs simulated annual maximum peak flows. Results are given for the period 1988-2005 and the Ourthe catchment at Tabreux. analysis in the Ourthe catchment.We estimate Q25 and Q100 with the simulated discharge series computed with the arithmetic KGE parameter set. We compute also the MAM7 with 5-year and 10-year recurrence intervals. We estimate MAM7(5) and MAM7(10) with the simulated discharge series computed with the logarithmic KGE parameter set. A Gamma statistical law has been fitted to annual maximum peak flows and MAM7 series to get the frequency values for the baseline period. Corresponding estimated discharges are indicated in Table 2. Whatever the index, the error amount of estimations made with simulated data is less than 10 %. Estimated quantiles of peak flows with observed data are slightly underestimated while estimated quantiles of low flows are well reproduced.

Figure 6. Observed vs simulated MAM7 values. Results are

given for the period 1988-2005 and the Ourthe catchment at Tabreux.

Table 2. Hydrological indices estimated with observed and simulated discharges.

	Observed data	Simulated data	Error
Index	m^3/s	m^3/s	%
MAM7(5)	1.5	1.5	0
MAM7(10)	1.2	1.3	8.3
Q25	369	340	-7.9
Q100	449	414	-7.8

So we are quite confident in the ability of the RR model to estimate peak floods and low flows frequency for the considered return periods.

The same calculation protocol as the one described above is applied to get hydrological indices from each of the 2006 discharge series simulated with transformed daily climate series.

5 RESULTS

5.1 Monthly statistical relationships between precipitation, air temperature and atmospheric pressure data

MLR coefficients fitted for the 1961-2005 period are presented in Table 3 for precipitation and in Table 4 for air temperature. The multiple correlation coefficients obtained for precipitation lie between 0.61 in August and 0.86 in February.

Except for May, the three predictors are always negatively correlated with precipitation meaning that precipitations are generally produced in low pres sure conditions in addition with, for winter months, advection of maritime air masses. For air tempera

ture multiple correlation coefficients are better and Table 3. Monthly coefficients of the multiple regression analysis (MLR) between precipitation and atmospheric predictors. Month r Intercept X of PGF Y of PGF Pressure J 0.80 2156.2 -4738.7* -4861.2** -2.1* F 0.86 1437.4 -2561.7** -6372.1* M 0.81 1436.1 -4818.8** -2.6** A 0.71 1556.8 -5145.5* -2.7** M 0.69 2676.3 4675.3* -4.6** J 0.67 10959.9 -10.7** J 0.75 11592.1 -8.7*/ -4.5** A 0.61 13278.0 -13.0* S 0.83 12119.4 -8456.3* -2363.8* -11.9* O 0.78 2458.6 -4983.9* -4.3** N 0.78 4469.6 -6552.8* -3919.1* -4.3* D 0.84 2531.1 -6284.1* -6347.8** -2.4* *Calculated from SLP data. **Calculated from Z500 data. Table 4. Monthly coefficients of the multiple regression analysis (MLR) between air temperature and atmospheric predictors. Month r Intercept X of PGF Y of PGF Pressure J 0.93 192.9 -217.6* -0.4*/0.4** F 0.92 190.3 -153.7* -0.4*/0.4** M 0.97 157 -140.9*/ -98.3*/ -0.4*/0.4** 94.6** 55.4** A 0.89 149.5 108.2* -0.3*/0.3** M 0.92 141.5 -117.3* -0.2*/0.4** J 0.94 56.6 241.2* 53.5** -0.2*/0.4** J 0.94 72.3 94.8** 122.5** -0.4*/0.3** A 0.92 187 186.4* 93.1** -0.4*/0.3** S 0.88 236.4 149.1* -0.4*/0.3** O 0.91 169.5 105.3* -69.1* -0.3*/0.3** N 0.87 213.2 -131.4* -0.4*/0.3** D 0.88 33.1 -208.8** -0.2*/0.3** *Calculated from SLP data. **Calculated from Z500 data. lie between 0.87 in November and 0.97 in March. In order to test the stationarity of the statistical linear relationships between atmospheric predictors and surface climate, they have been subject to a split-sample test. Results indicate similar explained variance levels over the validation period in comparison to calibration period (not shown). Therefore we are quite confident that the statistical relationships found for each month can be used to extend monthly precipitation and air temperature over the target period (1659–2100) with a reasonable level of uncertainty. 5.2 Estimated variability of the aridity index from AD 1659 to 2100 The aridity index has been calculated over the target period by using reconstructed and projected monthly values of precipitation and air temperature (Fig. 7). Some of the CMIP5 projections indicate drier conditions than the driest sub-period used to calibrate

Figure 7. 30-year moving average values of the aridity index from the Maunder Minimum (reconstructed) to the end

of the

21st century (projected according to 13 CMIP5 climate models forced with RCP 4.5 and 8.5 radiative scenarios). Dotted lines

represent one standard deviation of the mean. Results are given for the Ourthe catchment at Tabreux.

Figure 8. Time variability of precipitation from the Maunder Minimum (reconstructed) to the end of the 21st century (projected

according to 13 CMIP5 climate models forced with RCP 4.5 and RCP 8.5 scenarios).The ensemble mean of CMIP5 projections

is represented with a bold line. Results are given for January and the Ourthe catchment at Tabreux.

the RR model (Fig. 4).This means that in some

cases, whatever the selected calibration sub-period,

the model is applied in ungauged climate conditions

(extrapolation).

5.3 Estimated variability of monthly precipitation and air temperature from AD 1659 to 2100

By way of an example, Figure 8 and Figure 9 provide

the patterns of estimated precipitation time variabil

ity in January and the estimated air temperature time

variability in July from the Maunder Minimum (recon

structed) to the end of the 21st century (projected). For January precipitation, there is no evidence of a unique trend in precipitation change in the future. According to our calculation, projected range of changes is larger to the one experienced since the middle of the 16th century (Fig. 8). It 9means that January precipitation could reach an unprecedented level of low or high monthly totals during the 21st century. The mean scenario indicates a slight decrease of January precipitation during the next decades in comparison to the value of the baseline period (Fig. 8). For air temperature, after a multi-centennial period of stationarity, CMIP5 simulations lead to an unambiguous increase. At the end of the century, this rapid

Figure 9. Time variability of air temperature from the Maunder Minimum (reconstructed) to the end of the 21st century

(projected according to 13 CMIP5 climate models forced with RCP 4.5 and RCP 8.5 scenarios). The ensemble mean of

CMIP5 projections is represented with a bold line. Results are given for July and the Ourthe catchment at Tabreux.

increase could lead to a 30-year mean air temperature

up to 5 ∘ C higher to the baseline period value (Fig. 9).

The range of projected changes is much larger than the

reconstructed changes

5.4 Transformed daily precipitation and air temperature time series according to climate hazard assessment

Cumulative frequency distributions (CFDs) of trans

formed mean areal daily precipitation and mean areal

air temperature are shown in Figures 10a, b.A majority

of CDFs indicate a decrease in precipitation for daily

totals lying between 0 and 10 mm/day in comparison

to the baseline period (Fig. 10a). Highest daily totals

are expected to increase. Therefore, our methodology

of climate hazard likelihood tends to produce drier

climates as the most plausible climates for the future

in the study area. This does not however prevent an

increase of high daily precipitation during the year.

The impact of climate changes on the baseline CFD

of air temperature is quite important (Fig. 10b). The

CMIP5 projections being dominant in the range of pos

sible climate changes, it is not surprising to observe

a decrease of the frequency of frozen days. Most

of the possible climate changes indicate an increase

of air temperature of several degrees. Therefore, our

methodology of climate hazard likelihood tends to pro

duce warmer climates as the most plausible climates

for the future in the study area. This does not however

prevent the occurrence of low probability scenarios

of colder climates than today climate during the next

decades.

5.5 Changes in hydrological indices as a metric of catchment sensitivity to climate change

Remember that catchment sensitivity to climate

change is estimated through peak flows (Q25 and Q100) and lowest flows (MAM7(5) and MAM(10)). Looking at CFDs of 2006 discharges simulated with transformed climate series (Fig. 11), we can see that the baseline Q25 is exceeded 60% of the time (Fig. 11a) whereas baseline Q100 is exceeded 70% of the time (Fig. 11b). It means that Q25 and Q100 are expected to be higher than baseline period values when considering transformed climate series representative of climate hazard assessment. The time exceedance of baseline MAM7(5) and MAM7(10) values are strongly impacted by climate changes (Fig. 11c, d). As a consequence of dominant warmer and drier conditions in climate hazard assessment, present values are under-exceeded 70% of the time in the statistical distributions of MAM7(5) and MAM7(10) (Fig. 11). 5.6 Changes of significant hydrologic events as a result of climate change A possible valorization of our approach is to test the sensitivity of significant hydrological events of the observational period to a changing climate. As an example figure 12 focuses on the December 1993 flood event. According to the observed data, the discharge reached a maximum value of 370 m 3 /s which corresponds approximately to a Q25 value (Table 2). Using transformed precipitation and air temperature time series allows producing a large range of flood hydrographs with peak discharges ranging from 200 m 3 /s to 600 m 3 /s (Fig. 12). This represents a variation of ± 50% in comparison to the actual value. Figure 13 shows the low flow sequence of August 2003. Observed and simulated data indicate minimum low flow values of respectively 1.7 m 3 /s and 1.9 m 3 /s which are slightly superior to a low flow indice of MAM7(5). As a response to transformed climatic time series, for that specific minimum discharge value, calculated discharges range from 0.6 m 3 /s (−70% from the observed discharge) to 3.5 m 3 /s (+85% from the

Figure 10. Cumulative frequency distributions (CDFs) of mean areal daily precipitation (a) and mean areal air tempera ture (b) time series (2006 change factors) according to climate hazard assessment. The baseline CDFs are represented with black lines. Results are given for the Ourthe catchment at Tabreux.

observed discharge). Further works should investigate the change in the indices of the selected events in order to determine whether they may become common or rare in the future. Moreover, change in seasonal and monthly precipitation and air temperature have to be carefully examined to explain the hydrologic responses of the Ourthe catchment at Tabreux in a changing climate.

6 DISCUSSION AND CONCLUSION

We believe that the methodology presented in this paper allows the problem of climate change impact on water resources to be considered in a new per spective: by stress-testing the climate resilience of a

catchment through hazard assessment and catchment Figure 11. Cumulative frequency distributions of the relative changes (%) of annual maximum daily peak flows (a, b) and annual lowest mean discharge for 7 consecutive days (c, d) regarding to the baseline values. Changes are simulated with 2006 transformed climate series combining back and ahead knowledge of climate change. Results are given for the Ourthe catchment at Tabreux. sensitivity we produced a large number of plausible changes of extreme hydrological indices. This outcome is obtained by sampling a large spectrum of climate change factors computed through a jointly use of paleoclimate records and climate model outputs forced with extreme IPCC RCP radiative scenarios. For characterizing multi-decadal climate change, we developed and tested an empirical transfer function linking large scale atmospheric circulation and surface climate. This empirical transfer function is computed to generate long time series of monthly precipitation and air temperature representative of climate variability from the Maunder Minimum (natural climate change) to the end of the 21st century (anthropogenic

Figure 12. Observed and simulated hydrographs of the

flood event of December 1993. The black dotted line repre

sents the observed hydrograph. The black line represents the

simulated hydrograph with baseline climate data. Grey lines

correspond to the 2006 simulated hydrographs obtained with

transformed climate data series representative of past and

future possible climate changes.

Figure 13. Observed and simulated hydrographs of the low

flow event of August 2003. The black dotted line represents

the observed hydrograph. The black line represents the sim

ulated hydrograph with baseline climate data. Grey lines

correspond to the 2006 simulated hydrographs obtained with

transformed climate data series representative of past and future possible climate changes.

climate change). The main advantage of our statisti

cal downscaling approach is to use projected SLP and

Z500 pressure data which are recognized to be weakly

biased as well as in paleoclimate reconstruction expe

rience and in climate model experiment in compar

Byrd R.H., Lu P., Nocedal J., Zhu C. 1995. A limited

memory algorithm for bound constrained optimization.

SIAM Journal on Scientific Computing, 16, 1190–1208.

doi:10.1137/0916069.

Coron L., Andréassian V., Perrin C., Lerat J., Vaze J., Bourqui

M. & Hendrickx F., 2012. Crash testing hydrological mod

els in contrasted climate conditions : an experiment on

216 australian catchments. Water Resources Research, 48,

W05552.

Driessen T.L.A., Hurkmans R.T.W.L., Terink W., Hazenberg

P.,Torfs P.J.J.F., & Uijlenhoet R., 2010. The hydrological

response of the Ourthe catchment to climate change as

modelled by the HBV model. Hydrology and Earth System

Sciences, 14, 651–665.

Ernst, J., Dewals, B.J., Detrembleur, S., Archambeau, P., Er

picum, S., & Pirotton, M. 2010. Microscale flood risk

anal-ysis based on detailed 2D hydraulic modelling and

high resolution geographic data. Natural Hazards, 55,

181–209.

Kastendeuch P.P., 2007. Pressure gradient force, atmospheric circulation and climate in western Europe (1899-2002). International Journal of Climatology, 27, 15, 2055–2067.

Gupta, H.V., Kling, H.,Yilmaz, K.K. & Martinez, G.F., 2009. Decomposition of the mean squared error and NSE perfor mance criteria: implications for improving hydrological modelling. Journal of Hydrology, 377 (1–2), 8091. Luterbacher J., Xoplaki E., Dietrich D., Rickli R., Jacobeit J., Beck C., Gyalistras D., Schmutz C. & Wanner H. 2002. Reconstruction of sea level pressure fields over the Eastern North Atlantic and Europe back to 1500. Climate Dynamics, 18, 545–561, DOI 10.1007/s00382-0010196-6. Perrin C., Michel C. & Andréassian V. 2003. Improvement of a parsimonious model for streamflow simulation. Journal of Hydrology, 279, 275–289. Prudhomme C., Wilby R.L., Crooks S., Kay A.L. & Reynard N.S, 2010. Scenario-neutral approach to climate change impact studies: application to flood risk. Journal of Hydrology, 390, 198–209. Smith, J.B., K.C. Hallett, J. Henderson, and K.M. Strzepek, 2007. Expanding the Tool Kit for Water Management in an Uncertain Climate. Southwest Hydrology 6(1):26–27. Ullmann A., Fontaine B. & Roucou P., 2014. Euro-Atlantic weather regimes and Mediterranean rainfall patterns: present-day variability and expected changes under CMIP5 projections. International Journal of Climatology, 34, 2634–2650. DOI : 10.1002/joc.3864. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Efficient management of inland navigation reaches equipped with lift pumps in a climate change context

H. Nouasse, A. Doniec & E. Duviella

Mines Douai, IA, Douai, France

Université de Lille, France

K. Chuquet

Voies Navigable de France, Service de la navigation du

Nord, Lille, France

ABSTRACT: In a climate change context, less available water resource is expected in close future. The periods of drought will be more frequent with higher magnitudes. The quantification of the frequency, duration and intensity of future drought events has been proposed in recent publications. Inland navigation networks will be also impacted by these events. Hence, it is necessary to develop new infrastructure policy and to design adaptive water management strategies. One solution consists in equipping the inland navigation reaches with lift pumps able to pumping water back up to the upstream reaches. It is an interesting solution to limit the water consumption and to keep the water resource. However, the electricity costs should be very important. Thus, the main objective to this paper is to present an adaptive management strategy of inland navigation reaches equipped with lift pumps able limiting the electricity costs and guaranteeing enough water for the navigation accommodation. This management strategy is based on the definition of a quadratic optimization problem. An example of an inland navigation network that is composed of two reaches is proposed to detail the proposed approach and to highlight its performance.

1 INTRODUCTION

A historic agreement to combat climate change was adopted by 195 countries, the 12 th of December 2015 in Paris. The countries will pursue efforts to limit the temperature increase to 1.5 ∘ C from 2100 (Paris Agreement 2015). Even if this objective is reached, the Intergovernmental Panel on Climate Change (IPCC) expects a modification of the rain distribution over seasons and areas by the end of the century (IPCC 2013, IPCC 2014). RCP scenarios (Representative Concentration Pathways) on which future forecasts on temperature and rain have been generated and stud ies on the frequency and intensity of future flood and drought periods have been achieved and are avail able in the literature (Bates, Kundzewicz, Wu, & Palutikof 2008, Boé, Terray, Martin, & Habetsi 2009, Ducharne, Habets, Pagé, Sauquet, Viennot, Déqué, Gascoin, Hachour, Martin, Oudin, Terray, & Thiéry 2010). The recent study in (Wanders & Wada 2015) addresses the climate and Human impact on future hydrological drought across the world. Drought events are characterized by multivariate drought features in (Hao & Singh 2015). Other recent studies (Li, Huang, Ju, Lin, Xiong, Han, Wang, Peng, Wang, Xu, Cao, & Hu 2015) and (Park, Byun, Deo, & Lee 2015) are based on the most pessimistic RCP 8.5 scenario to quantify the frequency, duration and intensity of future drought events in several areas in China and Korea respectively. The hydrographical networks will be impacted by drought events, particularly the inland navigation networks(EnviCom 2008, IWAC 2009). The studies of the impact of climate change on inland navigation networks in UK (Arkell & Darch 2006) and in China (Wang, Kang, Zhang, & Li 2007) lead to the conclusion that the available water resource used to supply the networks will be insufficient during drought periods. Hence, it is necessary to propose adaptation measures allowing the system to function. To achieve this objective, it is first necessary to determine the resilience of the inland navigation networks against the drought periods, i.e. their capacity to allow the navigation. Secondly, adaptive management strategies have to be designed to optimizing the water resource management. Finally, it is often necessary to propose structural adaptations of the networks in order to improve their resilience. A multi-scale modeling approach of inland navigation networks is proposed in (Duviella, Horvàth, Rajaoarisoa, & Chuquet 2014) to reproduce their dynamics considering flood and drought events. According to this multi-scale model, a multiscale adaptive management strategy is designed in (Nouasse, Horvàth, Rajaoarisoa, Doniec, Duviella, & Chuquet 2016). It is based on tools allowing the optimal dispatching of water volumes amongst the

networks (Nouasse, Rajaoarisoa, Doniec, Chiron, Duviella, Archimède, & Chuquet 2015), and optimal algorithms to control the water levels (Horvàth, Rajaoarisoa, Duviella, Blesa, Petreczky, & Chuquet 2015). However, when the available water resource is insufficient structural adaptations of the networks have to be planed. It should be new storage areas, new gates, lift pumps. Lift pumps are used to back up the water to the upstream reaches. It is an interesting solution to keep the water resource. But, the use of lift pumps should cause important electricity costs. It is thus necessary to improve the adaptive management strategies considering the use of lift pumps by minimizing the electricity consumption.

In this paper, we propose an integrated model of inland navigation networks. It allows reproducing the dynamics of each navigation reaches that compose the networks. All possible interactions between the net work and its environment are taken into account. The dynamical variables are the water volumes on several hours. This integrated model is then used to design an optimal water resource dispatching method based on a quadratic optimization problem. The optimiza tion problem is defined to obtain an efficient water resource management with the lower electricity costs due to the water pumping.

The problem statement of water resource manage ment of inland navigation networks in a climate change context is detailed in section II. The proposed models, i.e. an integrated model and a flow network model, are described in section III. Section IV is dedicated to the formalization of the quadratic optimization problem. Finally, the case study that is based on the configura tion of a real inland navigation network is proposed in sectionV to highlight the effectiveness of the proposed method.

2 WATER RESOURCE MANAGEMENT OF INLAND NAVIGATION NETWORKS

In Europe, inland navigation networks cover large territories (see Figure 1). They are integrated in the Trans-European network (TEN-T 1) as environmental friendly transport mode solution (Mihic, Golusin, &

Mihajlovic 1993, Mallidis, Dekker, & Vlachos 2012, Brand, Tran, & Anable 2012).

In France, navigation schedules correspond gener ally to 14 hours each day of the week, with limitation on Sunday and closure a couple of days each year. During night, the navigation is not authorized with out exceptional agreements. The main objective of the inland navigation network managers is to gather all the conditions guaranteeing the navigation accommoda tion. To achieve this aim, it is necessary to control the water volumes in each parts of the networks. It con sists in supplying or emptying the network and then dispatching and balancing the water resource in all the

1 http://ec.europa.eu/transport/themes/infrastructure/ten-t

guidelines/index_en.htm Figure 1. European waterways 2 network taking into account the navigation demand and the climate hazards like flood and drought periods. These extreme climate events are expected more frequent and stronger in the future (Ducharne, Habets, Pagé, Sauquet, Viennot, Déqué, Gascoin, Hachour, Martin, Oudin, Terray, & Thiéry 2010). For example, in North of France, the results of the Explore2070 project 3 expect a decrease between 15 and 45 % of the available water resource from 2050 – 2070. Hence, the design of an efficient water management for inland navigation network becomes a challenging objective. An inland navigation network is mostly composed of several Navigation Reaches (NR). A NR is generally defined as the part of the network between two locks. The locks are operated to allow the crossing of boats. Close to the locks, the NR are often equipped with controlled gates that are used to control the water exchanges between NR and to guarantee the navigation conditions. The navigation can be accommodated only if the water level in each NR is inside the navigation rectangle (see Figure 2) and close to the Normal Navigation Level (NNL). Hence, the main management

objective consists in maintaining the level of each NR close to the NNL. The boundaries of the navigation rectangle are the High Navigation Level (HNL) and the Low Navigation Level (LNL). The water volume exchanges between NR are mainly caused by lock operations. When a ship is crossing a lock, a water volume empties the upstream NR and supplies downstream NR in few minutes. This water volume that depends directly from the size of the lock corresponds to several thousand of meter cubic. NR are generally located in the center of watershed. Thus, water exchanges with natural rivers are possible. Water intakes from natural rivers are used when the discharges from the controlled gates are not sufficient to keep the navigation conditions. It is also possible to reject the water in excess in natural rivers when the water volumes in the NR are too important. But, the 2
http://www.vnf.fr/vnf/img/cms/Tourisme_et_domaine
hidden/carte_france_europe_transport_deux_mille_onze_
201101041503.pdf 3

Figure 2. Navigation rectangle with high and low limits.

water exchanges with natural rivers have lower priority than the water exchanges between NR.

In some case, NR are equipped with lift pumps. They can be used to back up the water to the upstream NR. The electricity costs should be very important when lift pumps are used. Thus, using lift pumps can improve the water resource management but it requires to minimize the pumping cost. The proposed adaptive management strategy aims at guaranteeing the navi gation conditions by minimizing the pumping cost. It is based on an integrated model of inland navigation networks. This one is designed to represent their con figuration, to identify the water intakes and all the NR equipment, i.e. gates, locks and pumps.

3 INLAND NAVIGATION NETWORK MODELING

3.1 Integrated model

An inland navigation network is composed with a finite number η of interconnected NR. Its configuration can be represented by considering three elementary parts; linear NR, NR with a confluence, NR with a diffluence (see Figure 3.a). To distinguish the NR, they are numbered and denoted NR_i, with $i \in 1$ to η. Each NR_i is modeled as a tank that contains a volume of water, denoted $V_i(t)$, with t corresponds to a period of time. The volumes that correspond to the levels NNL, the HNL and the LNL are denoted such as $V_{LNL\,i} \leq V_{NNL\,i} \leq V_{HNL\,i}$. They are computed according to the geometrical characteristics of each NR_i. The management objective consists in keeping this condition: $V_i(t) = V_{NNL\,i}$, and at least the navigation conditions: $V_{LNL\,i} \leq V_i(t) \leq V_{HNL\,i}$.

Each NR_i are mainly supplied and emptied by controlled water volumes from locks and controlled gates (see Figure 3.b). Uncontrolled water volumes can also be considered such as withdrawals and supplies from water intakes, or exchanges with groundwater. However, it is generally very difficult to determine uncontrolled water volumes as deterministic values.

The set of controlled water volumes is:

• controlled volumes from the upstream NR that sup

ply the NR i , denoted V s,c i (s: supply, c: controlled), Figure 3. (a) Inland navigation network, (b) its integrated model. • controlled volumes from the NR i that empty the NR i , denoted V e,c i (e: empty), • controlled volumes from water intakes that can supply or empty the NR i , denoted V c i . These volumes are signed; positive if the NR i is supplied, negative otherwise, • controlled volumes from pump that can supply the NR i , denoted V s,p i (s: supply, p: pumped), • controlled volumes from pump that can empty the NR i , denoted V e,p i . The set of uncontrolled water volumes is: • uncontrolled volumes from natural rivers, rainfallrunoff, Human uses, denoted V u i (u: uncontrolled). These volumes are signed depending of their contribution to the volume V i (t) in the NR i . • uncontrolled volumes from exchanges with groundwater, denoted V g,u i (g: groundwater). These volumes are also signed. The dynamical volume of the NR i is computed according to the set of controlled and uncontrolled water volumes: with t corresponds to the current period of time and t − 1 the last one. The configuration of the network is taken into account for the computation of the controlled volumes. For a confluence, the controlled volumes coming from all the NR that are located upstream the NR i are added, without considering pumps. For a diffluence, those that empty the NR i correspond to the sum of the controlled volumes that supply the downstream NR. Controlled volumes from pumps are also computed according to the available lift pumps (see relation (2)). where� i gathers all the index of the NR that supply the NR i , i all the index of the NR that are supplied by the

Figure 4. Flow Network Model.

NR i , $ i the set of the index of the NR that supply the

NR i with pumps, and i the index of the NR supplied

by the NR i with pumps.

The efficiency of the proposed integrated model to

simulate real networks depends directly to the knowl

edge of each possible water volume contributions. In

this paper, some assumptions are considered to high light the performance of the proposed management strategy. Water exchanges with groundwater $V_{g,u\ i}$ are nonexistent, and the uncontrolled water volumes $V_{u\ i}$ are not considered. All the controlled water volumes are bounded with the known minimal and maximal daily volumes of water. The water volumes from locks depend on the daily average number of lock operations and on the volume of each lock. These data are known.

To describe the proposed water management strat egy, a flow network model is introduced. It is based on the integrated model.

3.2 Flow network model

A flow graph is defined as $G = (G_x ; G_a)$, where G_x is the set of nodes and G_a is the set of arcs. The set of nodes G_x contains the nodes representing each NR_i and two additional nodes, O and N, called respectively source and sink. The node O gathers all the water vol umes that supply the navigation network from natural rivers, the node N retrieves all the volumes of water from the navigation network. A directed arc is defined between two nodes as a couple $a = (i, j)$, with i the node that is leaving and j the node that is entering. For the sake of clarity, the name of the node corresponds to the index of the NR, i.e. $a = (NR_i, NR_j) = (i, j)$ or $a = (O, NR_j) = (O, j)$. On every arc inG_a, it is defined

a flow variable φ a , a ∊G a that can be expressed by φ i j
.

This flow varies with respect to capacities constraints

and demands relation on each node.

By considering navigation network depicted in Fig

ure 3 and four pumps (one between each NR), the cor

responding flow network model is shown in Figure 4.

The arcs are directed according to the configuration of the navigation network. The flows corresponding to the lift pumps are φ i i−1 , φ i i−2 , φ i+1 i and φ i+2 i . The particularity of the proposed flow graph is that each node NR i has a capacity denoted volume demand d(i) and expressed as: that must verify the following relation: The lower and upper bound capacities of the arc a, i.e. l a and u a , depend on the configuration and equipment of the networks. These capacities are expressed in volume. Without considering lift pumps, the set of the index of the upstream NR that are not supplied by another NR is given by !, and % is the set of the index of the downstream NR that not supply another NR. By considering examples in Figures 3 and 4, these sets are ! = { i − 2, i − 1 } , and %= { i + 1, i + 2 } . Thus: • upper bound capacities for arcs between two NR, i.e. { (i − 2, i), (i − 1, i), (i, i + 1), (i, i + 2) } in Figure 4, between O and NR j , j ∊!, i.e. { (O, i − 2), (O, i − 1) } , and between N and NR l , l ∊%, i.e. { (i + 1, N), (i + 2, N) } , are computed as the sum of the maximum available water volumes from water intakes (maximum positive V c i) and the required water volumes for the navigation, i.e. b ∊N the number of lock operations (V s,c i), • upper bound capacities for arcs between O and NR j , j ∉!, i.e. { (O, i), (O, i + 1), (O, i + 2) } , correspond to the sum of the maximum available water volumes from water intakes (maximum positive V c i), • upper bound capacities for arcs between N and NR j , j ∉%, i.e. { (i − 2, N), (i − 1, N), (i, N) } , correspond to the sum of the maximum water volumes that can empty the NR (minimum negative V c i), • upper bound capacities for arcs between two NR that correspond to lift pumps, i.e. { (i, i − 2), (i + 2, i) } depend directly to the maximal capacity of each pumps, • lower bound capacities for arcs between two NR, between O and NR j , j ∊!, and between N and NR l , l ∊%, are only the required water volumes for the b ∊N ships that cross the NR (V s,c i), • lower bound capacities for arcs between O and the NR j , j ∈!, and between N and the NR l , l ∈% are equal to 0, • lower bound capacities for arcs between two NR that correspond to lift pumps, i.e. { (i, i − 2), (i + 2, i) } are equal to 0. The flow graph being defined, it is used to design a optimal water management of inland navigation networks.

Figure 5. (a) Cost function corresponding to the volume

demands, (b) unitary lift pump electricity cost.

4 OPTIMAL MANAGEMENT OF INLAND NAVIGATION NETWORKS WITH LIFT PUMPS

The main objective consists in designing an efficient

water resource management of inland navigation net

works with lower electricity cost. This objective is

expressed as a quadratic criterion to minimize. To

define this criterion, a cost function is assigned to the

water volume in each NR. When the volume in the NR i

corresponds to V NNL i , i.e. d i (t) = 0, the cost is equal

to 0. But, more away this volume is from V NNL i , more

the cost is important. In this way, failure to respect

the navigation conditions is penalized. This cost func

tion is formalized according to a quadratic form (see

Figure 5.a):

withχ V = C max (V NNL i −V HNL i) 2 assuming that ❙ ❙ V NNL i −V HNL i ❙ ❙ = ❙ ❙ V NNL i − V LNL i ❙ ❙ , with C max the maximal cost, d n i (t) and

d d i (t) the demands during the night and navigation

periods.

The criterion C V (t) is minimized by dispatching the water on the network according to the navigation demand. In this way, water transfers between the NR using controlled volumes and lift pumps are used. The electricity cost for the lift pumps C P (t) is also taken into account. It is assumed that this cost is cheaper during night and more expensive during day, i.e. C n and C d respectively. It is expressed as a discontinuous function (see Figure 5.b). The pumping cost C P (t) is function of the pumped volumes:

where $ i is the set of NR that supply the NR i with pump, φ n i,j and φ d i,j the transfer volumes between NR i and NR j during night and day respectively. The quadratic criterion is defined by considering the period without navigation and the navigation period as: with η the number of NR, d n i (t) the demand of NR i during night, and d d i (t) during day, ! the set of the index of the upstream NR that are not supplied by another NR, % the set of the index of the downstream NR that not supply another NR. First terms of the criterion J V (t) correspond to the cost linked to the water volume in each NR, considering the period of night and the navigation period. Second terms are the cost due to the use of the lift pumps. Third terms aim at minimizing the water intakes from natural rivers, and last terms at minimizing the water releases in natural rivers. The minimization of the quadratic criterion has to verify equality constraints according to the configuration of the network. Thus, for the node NR i , i ∈ G x − { O, N } and for the night period, the equality constraint is defined as: with i the index of the NR that supply the NR i , and i the index of the NR that are supplied by the NR i without considering lift pumps, and d d i (t) the demand of the last period of time (navigation period). The same expression is used for the navigation period by inverting the exponents (n and d). The minimization of the quadratic criterion is carried out according to softwares such as the function fmincon 4 in Matlab. Then, it is possible to consider several scenarios and to study the efficiency of the proposed approach. 5 CASE STUDY The

considered inland navigation network is inspired from the one studied in (Nouasse, Rajaoarisoa, Doniec, Chiron, Duviella, Archimède, & Chuquet 2015). It is composed of 2 NR; NR 1 between the locks L 1 and L 2 and NR 2 between the locks L 2 and L 3 (see Figure 6.a). The gate G 12 allows controlling the water volume that supplies NR 2 with water from NR 1 . The integrated model of the network is depicted in Figure 6.b, with the controlled water volumes that can suply or empty each NR. NR 1 is supplied by operations of the lock L 1 with a controlled volume V s,c 1 (= V s,c,L 1). NR 2 is supplied by operations of the lock L 2 with a volume V s,c,L 2 , and by the gate G 12 with a volume V s,c,G 2 . Due to the configuration of the network, note that the volume from 4
http://fr.mathworks.com/help/optim/ug/fmincon.html?requestedDomain=www.mathworks.com

Figure 6. (a) Inland navigation network composed of 2 NR,

(b) the corresponding integrated model, (c) the corresponding

flow network.

Table 1. Physical characteristics of the NR in [m].

NR	Length	Width	Depth	HNL	LNL
1	10,000	50	4	+0.3	-0.3
2	15,000	50	3.5	+0.3	-0.3

NR 1 that supplied NR 2 , i.e. V s,c 2 , is equal to the vol

ume that empties NR 1 , i.e. V e,c 1 . The volume from NR 1

that supplied NR 2 , i.e. V s,c 2 , is also equal to the sum

of V s,c,L 2 and V s,c,G 2 . This difference is made to sim

plify the identification of the volume from the lock L 2

operation to the volume from the gate G 12 operation.

NR 2 is emptied by the operation of the lock L 3 with

the water volume V e,c 2 .

The gate G 1 (resp. G 2) allows supplying and emp tying the NR_1 (resp. NR_2) with the signed volume V_{c1} (resp. V_{c2}). To distinct if the volume is supplying or is emptying the NR, we introduce $V_{c,O1}$ (resp. $V_{c,O2}$) defined as the volume that supplies the NR_1 (resp. NR_2), and $V_{c,N1}$ (resp. $V_{c,N2}$) defined as the volume that empties the NR_1 (resp. NR_2). These volumes are distinguished to simplify the design of the flow net work model (see Figure 6.c), and to identify the flow conditions of each arc $\varphi_{i,j}$.

5.1 Study in a normal situation

The physical characteristics of the NR are given in Table 1. The water volumes in the two NR for the NNL are expressed in thousand of m^3 by: $V_{NNL1} = 2,000$ and $V_{NNL2} = 2,625$.

The navigation rectangle is defined according to the HNL and LNL (thousand of m^3): $1,850 \le V_{NNL1} \le 2,150$ and $2,400 \le V_{NNL2} \le 2,850$. Thus, the demand of each NR is bounded by: $-150 \le d_1(t) \le 100$ and $-225 \le d_2(t) \le 225$. A scenario is defined with the navigation schedules; the navigation is allowed from 6 am to 8 pm and stopped from 8 pm to 6 am (14 hours for navigation and 10 hours for non navigation, i.e. a navigation period $T_d = 50.4$ thousand of seconds and a night period $T_n = 36$ thousand of seconds).The controlled volumes for each lock are given in thousand of m^3 by: • $V_{s,c1} = 5 \times b$, • $V_{e,c,L1} = V_{s,c,L2} = 16 \times b$, • $V_{e,c2} = 4 \times b$, with b the number of lock operations during the navigation period. The water volumes from the controlled gates are known and bounded by: • G_1 : $[-2; 3]$ m^3/s, leading to $V_{c,O1} \in [0; 3T^*]$ and $V_{c,N1} \in [-2T^*; 0]$ (thousand of m^3), • G_{12} : $[0; 1]$ m^3/s, leading to $V_{e,c,G1} = V_{s,c,G2} \in [0; 1T^*]$, • G_2 : $[-5; 2.5]$ m

3 /s, leading to $V_{c,O2} \in [0; 2.5T^*]$ and $V_{c,N2} \in [-5T^* 0]$, where T^* is equal to T_d or T_n depending of the considered period. The bounds of the controlled water volumes lead to determination of the flow conditions of each arc of G, that are expressed in thousand of m^3 (see Figure 6.c): • $\varphi^d_{O1} \in [5 \times b; 5 \times b + 151.2]$; $\varphi^n_{O1} \in [5 \times b; 5 \times b + 108]$, • $\varphi^d_{O2} \in [0; 126]$; $\varphi^n_{O2} \in [0; 90]$, • $\varphi^d_{12} \in [16 \times b; 16 \times b + 50.4]$; $\varphi^n_{12} \in [16 \times b; 16 \times b + 36]$, • $\varphi^d_{1N} \in [0; 100.8]$; $\varphi^n_{1N} \in [0; 72]$, • $\varphi^d_{2N} \in [4 \times b; 4 \times b + 252]$; $\varphi^d_{2N} \in [4 \times b; 4 \times b + 180]$. The difference between flow conditions on night or navigation periods depend on the considered period T^* and also on the number of lock operations. During night, it is equal to $b_n = 0$, and during navigation period it is equal to $b_d = 12$. For example; $\varphi^n_{O1} \in [0; 108]$ and $\varphi^d_{O1} \in [60; 210.2]$. The maximum cost for the criterion $C_V(t)$ (see Relation 5) is $C_{max} = 2.000$ monetary unity. According to the proposed scenario, the quadratic problem is used to optimize the water volumes transfers between the two NR. The computed flows are shown in Figure 7.a for navigation period and in Figure 7.b during night period, by considering a normal situation. During day, a water volume of 192 thousand of m^3 is exchanged between the NR. It is due to the main navigation constraint on lock L_2. The lock L_1 is used to supply NR_1 with 60 thousand of m^3 and the gate G_1 with 132 thousand of m^3. The lock L_3 is used to empty NR_2 with 48 thousand of m^3 and the gate G_2 with 144 thousand of m^3. The demands in the two NR are equal to 0. Thus, the cost linked to the water volume differences in the two NR is $C_V = 0$.

Figure 7. (a) Scenario with b = 12 in a normal situation

during the navigation period, (b) during the night period.

Figure 8. (a) Scenario with b = 12 in a drought situation

during the navigation period, (b) during the night period.

5.2 Study in a drought situation

The second scenario consists in keeping the

same conditions but considering a drought period.

During drought periods, the available water resource

can decrease from 15 and 45 % at horizon 2050 – 2070

(data from Explore2070 project). A 40 % decrease of the available water resource from natural rivers

is considered. The two impacted controlled volumes

are G_1 : $[-2; 1.8]$ m^3/s and G_2 : $[-5; 1.5]$ m^3/s. That

leads to flows$\varphi^d_{O1} \in [5 \times b; 5 \times b + 90.72]$, and$\varphi^d_{O2} \in$

$[0; 75.6]$.

The quadratic problem is used again to optimize

the water volume transfers between the two NR. The

computed flows are shown in Figure 8 by considering

a drought situation.

During day, the navigation leads to a water volume

exchange of 192 thousand of m^3 between the two NR

thanks to the lock L_2 . The gate G_1 reaches its upper

boundary 90.72 thousand of m^3 ; the lock L_1 supply

ing NR_1 with 60 thousand of m^3 . However, it is not

enough to keep the NNL in the NR_1 . Thus, that leads to

a demand d^d_1 = 41.28 thousand of m^3 that corresponds

to 8 cm less than the NNL. There is no change for the

lock L_3 and the gate G_2 . During night, the gate G_1 is

opening to supply the NR_1 with 41.28 thousand of m^3

leading to a demand $d^n_1 = 0$. In this scenario, the cost

linked to the water volume differences in the two NR

is equal to C_V = 151.47. Figure 9. (a) Inland navigation network composed of 2 NR with one lift pump P_1 , (b) the corresponding integrated model, (c) the corresponding flow network. Figure 10. (a) Scenario of the inland navigation network equipped with a pump, with b = 12 in a drought situation during the navigation period, (b) during the night period. The last scenario consists in considering the same system equipped with a lift pump P_1 between NR_1 and NR_2 (see Figure 9.a). It allows the pumping of water from the downstream reach to the upstream reach with the volume V_{s,p_1} that is equal to V_{e,p_2} (see Figure 9.b). The pump has a capacity of 2 m^3/s. A new arc is added on the flow graph with flow condition: $\varphi^d_{21} \in [0; 100.8]$ and $\varphi^n_{21} \in [0; 72]$ (see Figure 9.c). The pumping costs are $C_n = 1$ and $C_d = 2$ in monetary unity, during night and navigation periods respectively. The proposed quadratic problem leads to the water volume transfers between the two NR depicted in Figure 10. In this case, the lift pump P_1 is used to back up a volume of 30.03 thousand of m^3. The demand is d^d_1 = 11.25 thousand of m^3 that corresponds to 2.25 cm less than the NNL. The solving of the quadratic problem aims at balancing the water volume transfer from the pump and the water demand to obtain the minimum cost. In this scenario, the cost linked to the water volume differences in the two NR is equal to $C_V = 11.25$, and the pumping cost is equal to $C_P = 60.06$. The total cost for this scenario is equal to 71.31, i.e. an economy of 47% compared to the system without pump. Moreover, it allows keeping the level close to the NNL.

6 CONCLUSIONS

A quadratic optimization problem is proposed in this paper to design an efficient management strategy of inland navigation networks equipped with lift pumps. It is shown that in a climate change context drought events could disturb the navigation particularly when less water volumes are available. It is necessary to propose a structural management of the inland navigation networks that consists in equipping it with lift pumps. The lift pumps are used to keep the water resource. However, it is also necessary to minimize the pumping electricity costs. The quadratic optimization problem leads to the optimization of the water volume manage ment with the lowest cost. The proposed approach is applied on an inland navigation networks composed of 2 reaches. Future works will consist in consider ing larger inland navigation networks, and proposing a predictive management optimization of the pumps to benefit to the cheaper electricity costs during night.

ACKNOWLEDGEMENT

This work is a contribution to the GEPET-Eau project (http://gepeteau.wordpress.com/enversion/) which is granted by the French ministry MEDDE – GICC, ORNERC and DGITM.

Arkell, B. & G. Darch (2006). Impact of climate change on london’s transport network. Proceedings of the ICE – Municipal Engineer 159, 231–237.

Bates, B., Z. Kundzewicz, S. Wu, & J. Palutikof (2008). Climate change and water. Technical repport, Intergov ernmental Panel on Climate Change, Geneva.

Boé, J., L. Terray, E. Martin, & F. Habetsi (2009). Projected changes in components of the hydrological cycle in french river basins during the 21st century. Water Resources Research 45.

Brand, C., M. Tran, & J. Anable (2012). The uk transport carbon model:An integrated life cycle approach to explore low carbon futures. Energy Policy 41, 107–124.

Ducharne, A., F. Habets, C. Pagé, E. Sauquet, P. Viennot,

M. Déqué, S. Gascoin, A. Hachour, E. Martin, L. Oudin,

L. Terray, & D. Thiéry (2010). Climate change impacts

on water resources and hydrological extremes in northern

france. XVIII Conference on Computational Methods in

Water Resources, June, Barcelona, Spain.

Duviella, E., K. Horvàth, L. Rajaoarisoa, & K. Chuquet

(2014). Multi-scale modeling approaches of inland navi

gation networks for their management in a global change

context. Transport Research Arena, Paris La Défense,

France.

EnviCom (2008). Climate change and navigation – water

borne transport, ports and waterways: A review of cli

mate change drivers, impacts, responses and mitigation.

EnviCom – Task Group 3. Hao, Z. & V. P. Singh (2015). Drought characterization from a multivariate perspective: A review. Journal of Hydrology 527, 668–678. Horvàth, K., L. Rajaoarisoa, E. Duviella, J. Blesa, M. Petreczky, & K. Chuquet (2015). Enhancing inland navigation by model predictive control of water level — the cuinchy-fontinettes case. Transport of Water versus Transport over Water – Exploring the dynamic interplay between transport and water – Carlos Ocampo-Martinez, Rudy Engenborn (eds). IPCC (2013). Summary for policymakers. Climate Change 2013: The Physical Science Basis. Contribution of Working Group I to the FifthAssessment Report of the Intergovernmental Panel on Climate Change – (eds) Stocker, T.F., Qin, D., Plattner, G.-K.,Tignor, M.,Allen, S.K., Boschung, J., Nauels, A., Xia, Y., Bex, V., Midgley, P.M. Cambridge University Press, Cambridge, UK and NY . IPCC (2014). Climate change 2014. The Core Writing Team, R. K. Pachauri and L. Meyer, Synthesis Report, https://www.ipcc.ch/pdf/assessment-report/ar5/syr/SYR_AR5_FINAL_full.pdf. IWAC (2009). Climate change mitigation and adaptation. implications for inland waterways in england and wales. Li, Y., H. Huang, H. Ju, E. Lin, W.

Xiong, X. Han, H. Wang, Z. Peng,Y. Wang, J. Xu,Y. Cao, & W. Hu (2015).Assessing vulnerability and adaptive capacity to potential drought for winter-wheat under the rcp 8.5 scenario in the huanghuai-hai plain. Agriculture, Ecosystems & Environment. Mallidis, I., R. Dekker, & D. Vlachos (2012). The impact of greening on supply chain design and cost: a case for a developing region. Journal of Transport Geography 22, 118–128. Mihic, S., M. Golusin, & M. Mihajlovic (1993). Policy and promotion of sustainable inland waterway transport in europe – danube river. Renewable and Sustainable Energy Reviews 15, 1801–1809. Nouasse, H., K. Horvàth, L. Rajaoarisoa, A. Doniec, E. Duviella, & K. Chuquet (2016). Study of global change impacts on the inland navigation management: Application on the nord-pas de calais network.Transport Research Arena, Varsovie, Poland. Nouasse, H., L. Rajaoarisoa, A. Doniec, P. Chiron, E. Duviella, B. Archimède, & K. Chuquet (2015). Study of drought impact on inland navigation systems based on a flow network model. ICAT, Sarajevo, Bosnie Herzegovia. ParisAgreement (2015). Adoption of the paris agreement—proposal by the president—draft decision -/cop.21. UNFCCC. Archived from the original on 2015-12-12. Retrieved 2015-12-12. Park, C.-K., H.-R. Byun, R. Deo, & B.-R. Lee (2015). Drought prediction till 2100 under {RCP} 8.5 climate change scenarios for korea. Journal of Hydrology 526, 221–230. Wanders, N. & Y. Wada (2015). Human and climate impacts on the 21 st century hydrological drought. Journal of Hydrology 526, 208–220. Wang, S., S. Kang, L. Zhang, & F. Li (2007). Modelling hydrological response to different land-use and climate change scenarios in the zamu river basin of northwest china. Hydrological Processes 22, 2502–2510. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Evaluation of changes storm precipitations during century for the

modeling of floods

V.V. Ilinich & T.D. Larina

Department of Meteorology, Hydrology and Water Flow Regulation Institute of Environmental Engineering of

Russian State Agrarian University, Moscow, Russia

ABSTRACT: Every dam of water reservoir must be projected in respect to catastrophic flood of normative

probability exceeding relative to maximum of water discharge and volume of flood. Some water reservoirs

are located on the small river basin. Usually we not have ranks observations for the runoff on small rivers.

Consequently there are different models and formulas for determination of flood characteristics in dependence

on storm rainfall. Different probabilistic quintiles of maximum daily precipitation are used in the models and

formulas. Some researchers have written hypothesis about increase of maximum daily precipitation (Groisman

2005, Dore 2005, Wilks 1999). However we must have very long time series of observation relative to daily

precipitation for the checking of the hypothesis. The research was dedicated to the checking of such hypothesis.

Several problems were solved in the work: selection of meteorological stations which have very long time series

of observation the daily rainfall; the forming statistical ranks for different discrete values of precipitation and

assessment their trends; determination of the main statistical characteristics relative to the formed ranks and

evaluation of their changes in the time; evaluation of the changes in the time of frequencies of appearance

dangerous storm rainfalls; evaluation of the changes in the time of probabilistic quantiles of maximum daily

precipitation in the frames of concrete distributions of random values. The results have showed that maximums

of daily precipitations are increasing during the time on majority of researched meteorological stations.

1 INTRODUCTION

1.1 Object of research and data of observation

The west part of the North Caucasus territory was taken

for the research. The territory had significant land scape change during last 30-40 years. In particularly, significant part of territory has been urbanized, usu ally this leads to increased summer temperatures and respectively to the increasing of the water vapor con tent in troposphere; the Krasnodar water reservoir has been created approximately 40 years ago and territory of rice fields was expanded, therefore an evaporation was increased; the total climate warming of region had place. So, water vapor content in troposphere increased because of different reasons last 30-40 years. Analog ical situation we can see in different region of World.

The research territory has more 20 meteorological stations, but only 4 stations have long time series of observations (not less 60 years) for the daily temper ature and precipitation. The stations are situated in Sochi, Armavir, Rostov, Elista cities (See Figure 1).

Here, storm rainfalls can be because of both atmo spheric fronts and of development of the air-mass clouds without frontal sections.

Majority documents in different countries and WMO recommends use statistical ranks of data observation not less than 30 years to determination hydrological and meteorological characteristics for flood control with help of water reservoirs. Accordingly, researches were made relative to three periods of data observations for every meteorological station: relative to first half of the period observation; to second of the period half and for all period (Sochi 1947-2006 years, Armavir 1947-2006, Rostov 1948-2007, Elista 1947-2010). So every period involves not less 30 years of observations. Three statistical ranks of dangerous precipitation have been formed for every period of data observation: a rank of maximum daily precipitation for every year; a rank of daily precipitation, which exceeded 30 mm; a rank of daily precipitation, which exceeded 60 mm (exception Elista and Rostov – third rank of daily precipitation has been formed on the base of values which exceeded 40 mm because the surrounding territory has more continental climate and is more dry, accordingly here almost there are not precipitation 60 mm and more). 1.2 Methods and materials The calculations of main statistical characteristics were performed for every statistical rank. Method of moments (Kristoforov 1988, Dmowska 2011) was used to obtain the following basic statistical characteristics: average of maximum daily precipitation (X 0), coefficients of variation (Cv) and skewness (Cs).Their numerical values are presented in Table 1.

Figure 1. Map of meteorological stations.

Table 1. Main statistical characteristics of the statistical ranks. The main statistical characteristics of the maximum daily precipitation for the year relative to periods of observations and their statistical errors Period of

City observations X 0 , mm ε average , % C v ε Cv , % C s ε Cs , %

Armavir 1947-1976 40.7 6.52 0.36 13.5 1.77 27.1 1977-2006 44.1 6.74 0.37 13.5 0.63 33.7 1947-2006 42.5 4.69 0.36 9.63 1.13 29.8 1947-1976 68.9 3.33 0.18 13.1 -0.1 -329.6

Sochi 1977-2006 74.4 3.21 0.17 8.64 -0.7 -62.9 1947-2006 71.7 2.37 0.18 9.26 -0.4 -80.4 1948-1977 34.5 6.97 0.38 13.5 1.23 38.7

Rostov 1978-2007 31.4 8.25 0.45 13.8 -0.56 -86.5 1948-2007 33.8 4.6 0.35 9.61 1.00 33.5 1947-1978 30.9 6.67 0.37 13.1 1.33 34.6

Elista 1979-2010 31.9 6.26 0.35 13.1 0.78 59.1 1947-2010 31.4 5.18 0.36 9.63 1.07 121

Results of table 1 show that Rostov is exception

where the average of maximum daily precipitation are decreasing relative to analyzed periods, however

variation has been increased significantly. The aver

ages of maximum daily values have increased during

last 3 decades on the all other meteorological stations.

Variation (Cv) had not change significantly. Statistical

errors of parameters X 0 , Cv and Cs were calculated

with help of formulas (Kristoforov 1988): where N = amount of members in statistical ranks. Statistical errors were obtained in allowed frames: less 10% for averages and less 15% for variation coefficients. Such accuracy is satisfactory relative to measurement of the daily river discharges which are occur under influence of storm rains. The third statistical parameter (coefficient of skewness – Cs) has significant statistical errors. The errors are significant even for 100-150 years of observation (Kristoforov 1988, Dmowska 2011) if we calculate Cs

Table 2. Values of skewness coefficients (Cs) and approximate ratios between Csand Cv. Parameters Period of

City observations C v C C s Calculated C s /C v Approximated C s /C v

Armavir 1947-1976 0.36 1.05 1.60 4.48 4.5 1977-2006 0.37 1.06 1.56 4.28 4.5 1947-1976 0.19 0.16 0.16 0.87 1.0

Sochi 1977-2006 0.18 1.02 0.04 0.22 1948-1977 0.39 1.06 1.55 4.08 4.0

Rostov 1978-2007 0.45 1.01 1.92 4.27 4.5 1947-1978 0.38 1.06 1.55 4.11 4.0

Elista 1979-2010 0.36 1.05 1.58 4.45 4.5

with help of classical method of moments with help of

next formula:

where σ = middle square deviation; N = total amount

of members in the statistical rank (i = 1 – N).

The formula (4) involves differences between observed values and their average raised in third level therefore relative error of skewness (Cs) is very significant.

Therefore there is offer for assessment of the asym metry in dependence on other statistical characteristics in frame some distributions of random values. The researches (Ilinich 2010, Ilinich 2014) have showed that there are dependences between Cs and other parameters, in particularly graph dependence from parameter “C” for concrete variation coefficients (Cv):

Where modular coefficient $K_i = X_i / X_o$.

So the skewness coefficients were determined with help the obtained dependences (Ilinich 2010, Ilinich 2014). Table 2 contains values of Cs and consequent ratios between Cs and Cv accordingly special gamma distributions of random values (Kritskiy 1981). The distribution is used in Russia and some other countries and has series of tables for different ratio of Cs/Cv with step 0.5 relative to their ratio. Therefore the table 2 contains two numerical values of Cs/Cv: the calculated and the approximated – nearest to the calculated value which has place in series of tables.

Values of table 2 were used for determination of modular coefficients Kp (Quantiles) from tables of special gamma-distribution of random values. Quan tiles of daily maximum precipitation (Xp) calculated:

Values of quantiles of daily maximum precipitation

(Xp) are represented in table 3. Table 3. Values of quantiles of daily maximum precipitation. Quantiles Period of City observations P = 1% P = 5% P = 10% Armavir 1947-1976 89.9 68.3 59.4 1977-2006 99 74.7 64.8 1947-2006 91.8 71.1 62.1 1947-1976 99.9 90.9 85.4 Sochi 1977-2006 104.9 95.2 90.7 1947-2006 102.5 92.4 88.2 1948-1977 76.9 59 51.4 Rostov 1978-2007 79.8 57.5 49.3 1948-2007 73.8 56.3 48.9 1947-1978 68.8 52.8 45.7 Elista 1979-2010 69.9 53.3 46.3 1947-2010 68.9 52.9 45.9 Figure 2. Chronological changes of the maximum daily precipitation for the year relative to period 1947-2006 years on meteorological station of city Armavir. The values of table 3 shows that almost all quantiles of daily maximum precipitation of last period exceed corresponding values of previous period with the exception of the Rostov values relative to quantiles 5% and 10%.Analogically we can conclude that majority of the quintiles of last period exceeds quintiles of the all time observation series. Chronological changes of the maximum daily precipitation relative to all data observation series are represented on the Figure 2-5.

Figure 3. Chronological changes of the maximum daily pre

cipitation for the year relative to period 1947-2006 years on

meteorological station of city Sochi.

Figure 4. Chronological changes of the maximum daily pre

cipitation for the year relative to period 1948-2007 years on

meteorological station of city Rostov.

Figure 5. Chronological changes of the maximum daily pre

cipitation for the year relative to period 1947-2010 years on

meteorological station of city Elista.

The trends of the graphs (Figure 2-5) have the raised

directions just on the graphs for Armavir and Sochi, this do not give sufficient bases for conclusion about

increase of dangerous storm precipitation during time.

Therefore other methods of analysis were applied.

Mann Kendall non parametric test (Mann 1945) was

used for estimation of trend of storm precipitation.

The procedure of estimation for the Mann Kendall test

(Mavromatis 2011) considers the time series of n data

points and X i and X j as two subsets of data where

i = 1, 2, 3, . . . , n − 1 and j = i + 1, i + 2, i + 3, . . . , n. Table 4. Values of statistics S and Zs. Meteorological stations Value S Value Zs Armavir 389 1.51 Sochi 471 1.82 Rostov 37 0.24 Elista 151 0.81 Each data value of time series is compared with all subsequent data values. If a data value from a later time period is higher than a data value from an earlier time period, the statistic S is incremented by 1. If the data value from a later time period is lower than a data value sampled earlier, S is decremented by 1. We have to keep next the order relative to calculation S (according to formulas 7 and 8): The variances (b 2) relative to S were defined by: Here − t i denotes the number of ties to extent i. The standard test statistic Zs was calculated as follows: Results are represented in table 4. In total, according to table 4 we can see that there is support of hypothesis about increasing of the trend relative to storm precipitation. Futher, the quantiles of maximum daily precipitation were researched. They determined with help of Weibull formula (Kristoforov 1988, Dmowska 2011) and are represented by empirical probabilistic points on the Figure 6-9. The Figure 6-9 give possibility to do conclusion, that majority of ordinates of empirical probabilistic curves for daily maximum precipitation of last period exceed corresponding values of previous period. The first and second statistical ranks involves dangerous cases of exceeding of daily precipitation which are equal 30 mm and 60 mm (for Rostov and Elista

Figure 6. Points of quintile values of empirical probabilistic

curves relative to meteorological station of city Armavir.

Figure 7. Points of quintile values of empirical probabilistic

curves relative to meteorological station of city Sochi.

Figure 8. Points of quintile values of empirical probabilistic

curves relative to meteorological station of city Rostov.

30 mm and 40 mm). Quantity of such cases on mete

orological stations are represented on Figure 10-17

in respect to equal duration of time series but differ

ent periods. Analysis of the graphs shows the total

increasing of cases of dangerous precipitations in

chronological development. Figure 9. Points of quintile values of empirical probabilistic curves relative to meteorological station of city Elista. Figure 10. Amount of dangerous cases of exceeding of daily precipitation 30 mm on meteorological station of city Armavir. Figure 11. Amount of dangerous cases of exceeding of daily precipitation 60 mm on meteorological station of city Armavir. 2 DISCUSSION An approach to the forming of the statistical time series of the extreme precipitation has own particularities, since several catastrophic values may appear during calendar year, but for the another year we may not observe any significant rain. It is necessary to notice –

Figure 12. Amount of dangerous cases of exceeding of daily

precipitation 30 mm on meteorological station of city Rostov.

Figure 13. Amount of dangerous cases of exceeding of daily

precipitation 40 mm on meteorological station of city Rostov.

Figure 14. Amount of dangerous cases of exceeding of daily

precipitation 30 mm on meteorological station of city Sochi.

Figure 15. Amount of dangerous cases of exceeding of daily

precipitation 60 mm on meteorological station of city Sochi. Figure 16. Amount of dangerous cases of exceeding of daily precipitation 30 mm on meteorological station of city Elista. Figure 17. Amount of dangerous cases of exceeding of daily precipitation 40 mm on meteorological station of city Elista. we do not have of the proved mathematical approaches for estimation of probabilistic characteristics extreme rains because their formation is complex process. Accordingly we can't to pick up a fairly objective method for forming a statistical number and appropriate distribution law of random variables at the present stage. We need new any hypothesis and a much longer period of observation of weather phenomena. 3 SUMMARY All research aspects prove that there is increase of the dangerous daily precipitations on majority of meteorological stations of the west part of the North Caucasus territory. Question about the correct mathematical assessment of the daily dangerous extreme precipitations in respect to their probabilities remains open. Statistical ranks of precipitation and maximum river discharges for the last 30 years can have quantiles which exceed significantly quantiles of full statisticals rank. This is necessary to take to consideration in the hydrological calculations. It is justified to use some non-standard methods for asymmetry evaluation in respect to meteorological and hydrological time series observations in frames of concrete distributions of random values.

There is no need to build analytical curves of prob

ability for definitions of quantiles of probabilities of

P = 5% and P = 10% depending on the parameters

Cv and Cs in the cases of presence of long meteo

rological observations (over 30 years). Such curves

are required for assessment of dangerous phenomena

for estimated quantiles of the distribution in limits

(P = 0.1% – P = 1%).

Groisman, P.Y. 2005. Trends in intense precipitation in the

climate record. J. Clim.: 18, 1326–1350.

Dore, MHI. 2005. Climate change and changes in global pre cipitation patterns: what do we know. Environ. In. 31(8):

1167–1181.

Ilinich, V.V. 2010. Assessment of skew in the frames of three

parametric gamma distribution. J. Prirodoobustroystvo,

No. 5 (In Russian): 71–75. Ilinich, V.V. 2014. Evaluation of asymmetry for ranks of extreme hydrological values. Proceedings of Conference: “21 century: fundamental science and technology III”, Vol. 3, North Charleston, SC, USA 29406: 10–13. Kristoforov, A.V. 1988. Theory of probability and mathematical statistics. Moscow: 128 (In Russian). Krtskiy, S.N. & Menkel, M.F. 1981. Hydrological basis of river flow management. Moscow: 255 (In Russian). Dmowska, Ed. R. & Hartman D. & Rossby H.T. 2011. Statistical methods in the Atmospheric Sciences. Inter. Geoph. Series Vol. 1. Oxford, OX51GB, UK: 668. Wilks, D. 1999. Inter annual variability and extreme-value characteristics of several stochastic daily precipitation models. Agril. AndForest Met. 93(3): 153–169. Mann, N.B. 1945. Non-parametric test against trend. Econometrica.13: 245–259. Mavromatis T. & Stathis D. 2011. Response of the water balance in Greece to temperature and precipitation trends. Theoretical and Applied Climatology 104: 13–24, doi 10.1007/s00704-010-0320-9. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Main impacts of climate change on seaport construction and operation

B. Koppe

Institute for Hydraulic Engineering, City University of Applied Sciences Bremen, Germany

ABSTRACT: Due to their location at the intersection between sea and land, marine facilities are most vulnerable

to various climate change impacts, but restrictions for the management of climate change challenges are evident

like differences in planning horizons and lack of relevant information. As a basis for climate change adapted

processes in port planning and operation a matrix

containing possible climate change impacts and possibly affected port assets is presented. Furthermore, steps of a vulnerability analysis of seaports against climate change effects are described and the sensitivity of specific port assets to climate change effects as well as possible adaptation are exemplified.

1 INTRODUCTION

Due to their location at the intersection between sea and land, marine facilities are most vulnerable to var ious climate change impacts. It is difficult to make general statements on the vulnerability of seaports to climate change. In some cases, already today climate change related problems in seaports are obvious, like changing properties of permafrost soils and related foundation problems in high latitudes. Other climate change impacts will influence port planning and opera tion only at a later stage, because in short and mid-term only minor effects are effective or the vulnerability of the system 'seaport asset' to the respective climate trend is only marginal. Additionally, ports located at the open sea have to bear other loads than estuary or lock-separated ports and the sensitivity of cargo handling with respect to wind and wave loads dif fers with cargo type. Therefore, a matrix containing possible climate change impacts and possibly affected port assets will be presented, which can be used by port planners and port authorities as a basis for climate change adapted processes in port planning and operation. Furthermore, examples of possible climate change adaptation measures in ports will be given.

2 RESTRICTIONS FOR THE MANAGEMENT OF CLIMATE CHANGE CHALLENGES TO SEAPORTS

Restrictions for assessment and management of climate change challenges to seaports are manifold as it is still difficult to reliably predict climate change effects and to deal with the long time spans climate change refers to. Some of the main restrictions are exemplified in the following. 2.1 Differences in planning horizons Climate change projections refer to time spans of decades to centuries because the climate variability makes it difficult to generate any short-term projections of less than 25 years. In contrast, today's port master plans are laid out for a time span of at the most 15 to 25 years, not least because industrial planning horizons seldom exceed 15 years. According to a survey representing 93 ports worldwide, most ports plan on a 5 to 10 years horizon (Becker, 2010). Therefore, many port planners perceive that impacts of climate change will not strike even their most far-reaching plans. This attitude neglects that already today some climate change effects are observable and that these changes will most probably continue within the next decades and centuries. Furthermore, the planning horizons in ports are generally much shorter than the design life of the port structures. Most marine structures are designed and constructed for a specific design life, which is taken to be its intended useful life, depending on the purpose for which it is required. The design life has to be defined for every single project on the basis of economical and financial considerations. Some generally recommended minimum design lives of port structures are listed in Table 1. For most port related infrastructure the minimum design life is around 30 to 60 years, whereas design lives of 100 or more years are assigned to port protection structures.

Differences in planning horizons of port master plans and selected port facilities are shown in Table 2. These figures indicate that structures built today have to withstand stresses not only in 20 but also in 60 or even 100 years. Today's port planning and investment decisions will affect how well the infrastructure accommodates the impacts of future climate change.

Table 1. Minimum design life of port structures according to [1] BSI, 1988; [2] Thoresen, 2003, [3] ROM, 1994, [4] AASHTO, 1993, [4] Witte, 2003.

Infrastructure	Lifetime in years
Quay walls	60[1]
Berth structures serving special industries (industrial planning horizons are often less than 15 years), container traffic, oil traffic etc. (specialized modern handling facilities lead to a relatively early outdating of berth facilities)	30[2]
Pavements	15–25[3], 30[4]
Open piers (open piled structures)	45[1]
Dry docks	45[1]
Breakwaters	60[1], 100[2]
Shore protection works	60[1]
Flood control	100[1], >100[2]
Superstructure	**Lifetime in years**
Superstructures	30[1]
Cranes (container cranes, multi	25[4]

purpose cranes)

Internal terminal transport (e.g. 10–15 4

straddle carriers, multi trailer trucks)

Outgoing transport – trains 30 4

Outgoing transport – trucks 10 4

2.2 Lack of relevant information

Climate models are founded on accepted physical prin

ciples and are able to reproduce observed features

of current and past climate changes. They can pro

vide credible quantitative estimates of future climate

change especially at continental and global scales.

During the last years, there has been a significant

progress in the understanding of e.g. sea level change.

Nevertheless, there are still significant uncertainties,

particularly related to the magnitude and rate of the

ice-sheet contribution for the 21 st century and beyond,

the regional distribution of sea level rise, and the

regional changes in storm frequency and intensity

(IPCC, 2013).

The use of global scale results for regional plan

ning is questionable as climate as well as most climate

change processes vary from region to region. This

variation is driven by ocean dynamical processes,

movements of the sea floor, and changes in gravity

due to water mass redistribution (land ice and other terrestrial water storage) in the climate system (IPCC, 2013).

For port planning and operation, climate change information on regional and local scale is required but often not available in requested quality. In addition, interactive processes of climate, marine and geologic systems will lead to regionally variable secondary changes, which have to be observed and assessed at a regional scale. In consequence, not only climate scien tists but also experts from other disciplines have to be included in order to deliver reliable regional scale data

for port and infrastructure planning and maintenance. Table 2. Differences in planning horizons and lifetime of port facilities. 2.3 Treatment of uncertainty Hydraulic, geotechnical, morphological and structural processes are dynamic and stochastic in nature and their interaction is often quite complex. Since the mid-nineties of the 20th century probabilistic design tools have been developed in the field of port planning. Nevertheless, even today mainly deterministic approaches based on discrete planning parameters are in use. Deterministic values are not only applied to describe design loads due to water levels, waves, currents as well as dead and life loads. Also economic and traffic forecasts are transferred into deterministic planning parameters. On contrary, climate change statements are always given in probabilistic terms using likelihood ranges. Climate processes are extremely complex, interactive and sensitive. Therefore, it is hard to make any certain climate change statement and projection uncertainties as well as changes in predictions will have to be accepted not only today but also in future. Consideration of climate change in planning processes always means the need for a dynamic decision process that can adapt to new information, and accommodate feedback. It is important to consider a range of plausible scenarios in port planning and to value flexibility in order to assess the cost-effectiveness of flexible options. 2.4 Financial aspects The port related added values comprise not only direct economic benefits relevant mainly for the private sector but also additional indirect benefits like enhancement of trade, increases in production volumes and trade-related services. The installation and operation of a port will often result in an economic transformation of a whole region. These port-related economic multiplier effects are the main reason for direct public sector investments in ports.

Table 3. Aspects of a sensitivity analysis.

1. System of interest, e.g. breakwater, quay wall, flood protection, terminal infrastructure and superstructure, drainage system, storage area, hinterland connection

2. System planning parameters, e.g. functional and structural design, operation characteristics, lifecycle aspects, state-of-the-art aspects

3. Current relevant climate conditions affecting the system in the planning area, e.g. cold and long lasting winters, hot summers, often and intensive storm surges, heavy precipitation

4. Existing stresses under current climate conditions, e.g. coastal erosion problems, high sedimentation rates, flooding, extended operation downtime due to wave action, limitations to port operation due to high algae growth rates in spring and summer

5. Projected change of relevant climate conditions (including climate conditions that will become relevant due to projected changes) in the short-, midand long-term, e.g. rise of air and water temperature, more intense storm surges, sea level rise, changes of wind direction distribution

6. Projected impacts of climate change without preparedness action, e.g. foundation and structural problems, increase in port operation down-time, lack of terminal or storage space, increase in flooding damages

7. Degree of system sensitivity to climate change, e.g. high, moderate, low, uncertain

In many ports today, the public sector mainly acts as

planner, facilitator, developer, and regulator providing

maritime access and hinterland connection, whereas the private sector acts mainly as service provider and operator and sometimes also as developer.

Depending on the port model, financial decisions have to be made by the public or private sector alone or in agreement of both parties with investments contributed by both sides. Public investments, either originating from state or district government or munic ipality, depend strongly on political and economic policies, whereas private investments rather reflect the assumption of direct profitability.

Long-living infrastructures like breakwaters, locks and coastal protection works are often expensive and cost recovery cannot be achieved in normal long-term loan repayment periods of 20 or 30 years. Therefore, private investments for these works are rare and pub lic investments are needed, often granted under the assumption of substantial enhancement of regional and/or national economies.

Considering limited financial resources, the approval of costly investments in ports is often hard to achieve and the investments have to be justified by economic data. This conflicts with the fact that climate change predictions are still imprecise especially on regional scales. Nevertheless, adaptation issues have to be taken into account particularly in the planning of long-living climate-sensitive investments, which are most vulnerable to climate change effects due to their long lifetime.

3 CLIMATE CHANGE SENSITIVITY AND ADAPTIVE CAPACITY ANALYSES FOR SEAPORTS

It is difficult to make general statements on the vulner ability of seaports to climate change. In some cases, already today climate change related problems in seaports are obvious, like changing properties of per mafrost soils and related foundation problems in high latitudes. Other climate change impacts will influ

ence port planning and operation only in a later stage because in short and mid-term 1 only minor effects of climate change are effective or the vulnerability of the system ‘seaport’ to the respective climate trend is only marginal. Additionally, ports located at the open sea have to bear other loads than estuary or lock-separated ports and the sensitivity of cargo handling with respect to wind and wave loads differs with cargo type. Vulnerability can be described by three components: exposure, sensitivity and adaptation capacity. The determination of the exposure of a specific port has to be based on an analysis of climate change impacts on a regional scale.Then, a sensitivity analysis and an adaptive capacity analysis have to be conducted for the port as a whole or for particular port assets. Combining the findings of these analyses will result in a quantification of the vulnerability of the seaport to climate change impacts. 3.1 Sensitivity analysis A sensitivity analysis will examine the aspects listed in Table 3. 3.2 Adaptive capacity analysis An adaptive capacity analysis will address the issues listed in Table 4. 4 CLIMATE CHANGE IMPACTS ON SEAPORTS Due to their location at the intersection between sea and land, marine facilities are most vulnerable to changes of all water related parameters like mean relative sea level, storm water levels, wind waves and swell, tidal regime, sedimentation rates, waterborne immigration of species,

water salinity and acidity. Furthermore, seaports can also be affected directly by temperature, precipitation and wind
1 Short-term: up to 15 years, mid-term: up to 30 years, longterm: up to 100 years

Table 4. Issues of an adaptive capacity analysis.

1. System of interest, e.g. breakwater, quay wall, flood protection, terminal infrastructure and superstructure, drainage system, storage area, hinterland connection

2. System planning parameters, e.g. functional and structural design, operation characteristics, lifecycle aspects, state-of-the-art aspects

3. Ability of the system to cope with climate change effects, e.g. changes in the tidal regime accompanied by stronger tidal currents may lead to scour deepening at existing quay walls, yet being in a range not affecting the structural strength of the construction

4. Obstacles to a system's ability to cope with climate change effects, e.g. – Legal regulations, e.g. national borders limiting a required port extension/relocation to adopt to new climate situations; environmental regulations restricting dredging works required to balance higher sedimentation rates due to changes in tidal regime in an estuary – Limited management flexibility due to a high number of competing requirements, like navigation and port services, flood control, irrigation supply, water quality, protection of endangered species, cooling purposes, recreation – Limited management flexibility due to geographic situation, e.g. relocation in case of significant sea level rise may not be possible because of a hilly hinterland

5. Existing stresses and constraints limiting the system ability to accommodate climate changes, e.g. existing overload with respect to e.g. structural stability or flooding levels, lack of terminal or storage space, lack of financial means

6. Effectivity and success of efforts already undertaken to deal with climate changes (or other challenges), e.g. analysis of structural weak points, managing plans

changes with respect to e.g. empty container storage

heights, terminal pavement durability and stormwa ter drainage system capacity, cooling system energy demands etc.

In Table 5 selected phenomena of climate change and their possible opportunities and risks for seaports are described. In some cases a potential improvement or worsening of the situation can only be assessed in a case-by-case manner. For example, changes in the wind direction distribution may reduce or increase sedimentation in the port access channel and in harbor basins. Furthermore, a change of one parameter due to climate change may also result in changes of other parameters not directly linked to climate change impacts. For example, an increase in sea level due to climate change potentially enlarges the water depth in an estuary serving as a port access but also influences the tidal regime, which may lead to larger quantities of sediment in the access channel and eventually smaller water depths. Often quite complex process chains have to be considered in the assessment of climate change impacts.

5 CLIMATE CHANGE SENSITIVITY OF PORT ASSETS AND ADAPTATION MEASURES

Accurate predictions of regional climate change impacts on design and operation of seaports are currently not available and will be difficult to attain in future. Nevertheless, as constructions designed today may be subject to changing loads due to cli mate change during their lifetime, considering climate change predictions is essential already now. Design loads like extreme water levels and wave heights are often defined by statistical return periods, e.g. 50-year wave height or 100-year high water level. Cli mate change impact may change a former 100-year return period to a 70or 40-year-return period with the effect of a significant increase in the failure risk of the construction. Therefore, consideration of climate change effects in port planning and investment schedules should rather start now than tomorrow. As accurate regional climate change predictions are not yet available, probable scenarios have to be defined as a planning tool for today's port construction and future port operation. The adaptation of seaports to climate change requires proper vulnerability analysis based on sensitivity and adaptive capacity analyses of specific port systems referring to locally anticipated climate change effects. The following four port asset categories can be distinguished: – Basic port infrastructure like maritime access channel, port entrance, protective works and hinterland connection – Operational port infrastructure like inner port channels and port basins, quay walls and port internal traffic systems – Port superstructure like pavements, drainage system, stacking areas, tank farms, silos, and warehouses – Port equipment like ship and shore handling equipment, as well as cargo handling and storage equipment Within the present paper, general remarks will be given to the port assets maritime access channel, breakwaters and jetties, and hinterland connections by roads. 5.1 Maritime access channel The design of the maritime access channel regarding length, width and run is strongly dependent on the port location with respect to open sea, estuary or waters with large tidal range. Often, water depth in the port access channel is not sufficient for larger vessels entering the port and the required water depth has to

Table 5. Selected phenomena of climate change and possible impacts on seaports.

Phenomenon and direction of

trend Possible impacts on seaports

Temperature

Increase in annual mean air

temperature Risks: More high temperature related downtimes (low latitude zones) Decline of extent of permafrost soils leading to foundation and erosion problems (high latitude zones) Immigration of new species potentially leading to higher deterioration rates of port infrastructures and problems in port operation

Increase in annual mean water

temperature Opportunities: Less ice related downtimes (high and middle latitude zones) More ice-free ports and longer shipping seasons in high latitudes Risks: Growth rate increases of waterborne species Immigration of new species potentially leading to higher deterioration rates of port infrastructures and problems in port operation

Increase in extreme high air

temperatures Risks: Increase in power consumption for cooling systems (e.g. reefer containers and refrigerated warehouses) Higher deterioration rates of pavements (e.g. terminal surfaces) Increase in stress on temperature sensitive structures made of metal (e.g. container handling cranes, warehouses) Increase in heat related disruptions in the hinterland traffic

Increase in extreme high water

temperatures Risks:Restrictions to port related operations (e.g. maintenance dredging) due to low oxygen content in warmer water Increase in production downtime in port located industrial plants with water cooling systems due to exceedance of water temperature threshold values

Decrease in number of days

with sub-zero temperatures Opportunities: Less low temperature related downtimes (high and middle latitude zones) More ice-free ports and longer shipping seasons in high latitudes Risk:Decline of the extent of permafrost soils leading to foundation and erosion problems (high latitudes)

Precipitation

Increase in intensity of heavy

rainfall events Risks: Increase in heavy rain fall related port operation downtime Increase in flood risk due to high water levels in the hinterland Increase in stormwater drainage system capacity overload Increase in deterioration rates of stormwater drainage systems Increase in stormwater related disruptions of the hinterland traffic

Increase in the amount of

annual rainfall Opportunity: Improvement of inland navigation conditions during low water season due to higher discharge rates and consequently higher water levels in inland rivers Risks: Worsening of inland navigation conditions during high water season due to higher discharge rates and consequently higher water levels Worsening of inland navigation conditions during high water season due to higher discharge rates and consequently less overhead clearance below bridges Worsening of inland navigation conditions during high water season due to increased flow velocities caused by higher discharge rates Higher water content in soils and subsequently slope stability problems

Decrease in the amount of

annual rainfall Opportunity:Reduction of inland navigation downtime related to high water levels or high discharge rates Risks: Worsening of inland navigation conditions during low water season due to lower discharge rates and consequently lower water levels in inland rivers Worsening of water quality in inland rivers during low water season due to lower discharge rates; consequently possible restrictions in maintenance works

Wind

Increase in frequency of storm

events Risks: Increase in wind related port operation downtime Increase in high water level related port operation downtime Increase in wave related port operation downtime Increase in storm related disruptions in the hinterland traffic Increase in coastal erosion rates especially at sandy coasts (continued)

Table 5. Continued

Phenomenon and direction of

trend Possible impacts on seaports

Increase in intensity of

extreme storm events Risks: Increase in storm surge related damages to port infraand superstructure Increase in port operation downtime related to infraand superstructure repair Decrease in allowable storage heights, especially on empty container storage yards – increase in required storage area Increase in storm related disruptions of the hinterland traffic Increase in coastal erosion rates esp. at sandy coasts

Change of wind direction

distribution Opportunities: Potential decrease in water level, wind, wave related stresses as well as sedimentation rates, depending on the individual case Risks:Potential increase in water level, wind, wave related stresses as well as sedimentation rates, depending on the individual case

Sea level

Increase in mean relative sea

level Opportunity: Increase in water depths in port access channels and harbor basins Risks: Increase in flood levels in extreme events (storm surges) Increase in coastal erosion rates especially at sandy coasts Increase in wave stresses at quay walls, breakwaters, extended wharfs and other port infrastructure during mean and high water levels Decrease in overhead clearance below bridges and port cranes

Changes of the tidal regime

in estuaries due to changes in

mean relative sea level Opportunities: Increase in water exchange rate and consequently increase in water quality in case of higher tidal range Potential increase in the erosive potential of the ebb stream velocities relative to flood stream velocities in case of higher tidal range resulting in less sedimentation Risks:Potential erosion around bridge footings in case of higher tidal range and subsequent higher current velocities Temporarily higher sedimentation rates due to upstream shift of the tidal boundary and the brackwater zone Potential long-lasting increase in sedimentation rates due to tidal pumping effects (increasing flood stream velocities relative to ebb stream velocities) Immigration of new species potentially leading to higher deterioration rates of port infrastructures and problems in port operation

Water chemistry

Increase in salinity Opportunity: Potential decrease in growth rate of domestic species Risks: Increase in the water's corrosive potential to steel and con-crete structures Potential immigration of new species Potential increase in growth rate of domestic species

Decrease in salinity Opportunities: Decrease in the water's corrosive potential to steel and concrete structures Potential decrease in growth rate of domestic species Risks: Potential immigration of new species Potential increase in growth rate of domestic species

Acidification Risk: Increase in the water's corrosive potential to steel and concrete structures

Species

Increase in growth rate of

domestic species Risks: Potential increase in harmful species population leading to deterioration of construction material High amount of organism in conjunction with lack of oxygen leading to high sedimentation rates

Immigration of new species Risk: Potential increase in harmful species population leading to deterioration of construction material

be guaranteed by rock drilling and blasting or dredg

ing (both investment and maintenance dredging) as

well as supporting constructions like groins and train

ing structures, the latter being mainly used in estuary

ports.

Two general categories summarize key characteris tics of maritime access channels: – Channel layout, i.e. plan view with straight and curved sections – Channel cross section characterized by depth, width, side slopes, and lateral restriction or nonrestriction In general, the layout and dimension of maritime access channels are determined as multiples of the

design ship's beam and draft, taking into account

additional factors like:

– Vessel traffic characteristics, e.g. traffic mix and traffic density with respect to one-way or two way traffic; vessel's length; vessel's velocity with respect to squat; overall vessel maneuverability; dangerousness of transported goods

– Environmental factors, e.g. tide, wind, waves, and currents

– Structures, e.g. bridges with respect to their clear ance height as well as lateral constructions like piers, training structures, and groins

In estuaries climate change risks with respect to maritime access channels are expected mainly due to an increase in sedimentation and thus a decrease in water depth. Changes in sediment transport are gen erally difficult to predict and countermeasures may be designed rather in a reactive manner and hardly in advance. Shorter maintenance dredging intervals are the most common countermeasure. Depending on the result of a detailed investigation of the hydrology and sedimentology, the construction of training structures influencing currents may be useful.

The entrance channel of ports at the open sea is generally less vulnerable to climate change effects as sea level rise and resulting larger water depths bear more opportunities than risks.

5.2 Breakwaters and jetties

Breakwaters and jetties border the port basin and entrance from the sea or estuary serving as a protection against waves and currents. In some cases, the inner side of the construction may be used as a berthing place.

The orientation and width of the entrance open ing between the breakwaters and jetties must fulfill two opposing demands, which have to be carefully balanced:

- Considering navigational aspects, the port entrance should be as wide as possible
- For protection of the inner port and the entrance area against waves, currents, and sedimentation the port entrance should be as narrow as possible

The entrance should be preferably oriented in such a way that prevailing winds blow towards or oppo site entering vessels as transverse winds and waves may create difficult navigational conditions. Entrance jetties should not end in the highly turbulent zone of

breaking waves.

Different types of breakwaters or jetties can be

distinguished as follows:

- Sloping or rubble-mound breakwater

- Vertical breakwater

- Composite breakwater

- Special breakwaters like piled or floating breakwa

ters which are suitable in seaports only to a limited

extend due to their high wave transmission ratio Generally, the following loads have to be assessed within the design process of sloped or vertical breakwaters: - Design water depth - Design wave parameters like wave height, wave period and wave angle relative to the construction Depending on type, shape and transmissibility of a breakwater, wave forces, wave run-up and overtopping as well as wave transmission and reflection have to be assessed to determine the breakwater's height and layout. The latter should consider armor unit weight and layer thicknesses in case of rubble mound breakwaters and overall stability in case of vertical breakwaters. Looking at overtopping rates a simple arithmetic example can show the problem. According to EurOtop 2007, the mean overtopping rate at a vertical structure with a crest freeboard of $R_c = 1.5$ m and a design wave height of $H_s = 1.0$ m is $q = 2.5$ l/(s*m). Considering a sea level rise of +0.5 m the mean overtopping rate would increase to $q = 9.3$ l/(s*m) and to $q = 34.1$ l/(s*m) for a sea level rise of +1.0 m, respectively. The increase in mean overtopping rates may result in significant utilization restriction or even structural damages. As breakwaters are long-living and costly structures, the consideration of anticipated climate change effects is highly recommended in the design of new structures. Increasing wave loads have to be considered in the determination of armor unit weights of sloped constructions or component weight and stability of vertical constructions. It is recommended to conduct a sensitivity analysis of possible incident wave heights and periods dependent on wave direction to check if possible changes in the wind direction distribution may lead to significant increases of wave loads in the port entrance and at terminals. Rising water levels have to be encountered by heightening the crest level or by arrangements offering a later heightening of the crest level, like wider bases and stronger foundations. 5.3 Hinterland connection by roads Hinterland transportation consists of a mix of road freight, rail, and inland waterway transport. As hinterland connections play a crucial role in the freight network of seaports, it is of prime importance to offer a high productivity and a permanent operability. The productivity is strongly dependent on the number of traffic lanes and railtracks with respect to roads and rails and on the width and depths of the fairway with respect to inland waterways. The oper-ability depends on the stability and resistance of the construction material against loads as well as on sedimentation processes in waterways. Road infrastructure is a long-lived investment. Roads typically have design lives of 20 to 40 years and bridges of 100 years. Road maintenance works include cleansing and restoring of surface and sub-surface drainage systems as well as surface rejuvenating.

Road constructions typically comprise a number

of layers, consisting at least of surface layer, bed

ding layer, road-base, sub-base, capping layer, and

plane. The materials of the surface layer as well as

the road-base are generally bound in a less permeable

matrix such as cement or bitumen. The layers of the

road foundation can be made up of unbound granular

material. Road pavements made up with a bituminous

surface are flexible, whereas pavements made up of

high strength concrete or reinforced concrete are rigid.

Mainly asphalt or cement is used as a matrix for the

upper layers as these materials are reliable and strong.

Nevertheless, both materials have their advantages and

disadvantages. One main difference is the durability,

where concrete roads are more durable resulting in a longer life compared to asphalt roads. Concrete roads are less sensitive to oil leakages than asphalt layers and extreme weather conditions like heat and frost will result in lesser damage. Therefore, the mainte nance cycle of concrete roads is longer than for asphalt roads. On the other hand, asphalt roads are easier to repair because a partly repair is possible as well as a re-layering. Furthermore, asphalt roads provide better safety for vehicles because of better skid resistance and provision of good traction. The choice of the surface material in a specific project requires a consideration of all of these aspects.

Roads connecting the hinterland of ports are loaded by heavy goods vehicle traffic as well as climatic con ditions like severe frost, freeze-thaw cycles, excessive rainfall or extreme heat. Direct weather impacts on surface material are the following:

– High temperatures, both maximum temperature as well duration of high temperatures, are significant especially with respect to bituminous pavements leading to adverse effects on the hardness of the road surface, fatting up of the road surface as well as thermal expansion and contraction affecting the integrity of the road surface resulting in potholing, rapid loss of surface condition, and rutting

- Cycles of freezing and thawing can cause volume changes resulting in disaggregation effects within the road structure; climate change may even increase these effects because rising temperatures does not exclude the possibility of a greater extent of thawing and refreezing, possibly following a day and night pattern
- Changes in rainfall patterns can alter water balances and influence pavement deterioration
- Changes in temperature and rainfall patterns may interact where higher temperatures increase cracking, which amplifies the effects of increased rainfall
- Melting permafrost may lead to a destabilization of road beds

Beneath material aspects also road elevation plays an important role in port operation as flooding may close the transport route and lead to downtime. Higher mean sea levels and storm surge water levels as well as more intense wave exposure may lead to an increase in flood risk. Most climate change impacts bear risks for roads. Considering climate change issues in road planning potentially will lead to significant cost reductions on the long run, allowing for longer maintenance intervals and longer lifetime of the structure as well as less downtime. Dependent on the project area, a more sophisticated road-bed, a more resistant road surface – e.g. made of concrete instead of asphalt – and a higher road elevation may be necessary to cope with climate change effects. Special attention must be given to thawing processes of permafrost soils as permafrost serves as a foundation for most structures like roads, railways and buildings in the Canadian Arctic, Alaska, and Siberia. With the loss of permafrost, efforts and expenses for secure foundations in these regions will raise significantly. 6 CONCLUSIONS Seaport infrastructure in general is characterized by long life spans, enforcing long-term planning horizons. Climate change issues may impact the construction and operation of seaports in manifold ways, with respect to navigation water levels, overhead clearances, sedimentation patterns, structural loads, material deterioration, etc. Therefore, early involvement of climate change aspects, especially in port planning processes of long living assets, is highly recommended. In view of the immense damage potential of climate change impacts on port structures, an understanding and a consideration of the expected impacts of future climate change by port planners, port authorities, terminal operators and asset managers allows for considerable long term cost savings. At the broad strategic level, forewarning experts and authorities will allow them to better prepare to deal with costly future effects. Uncertainties in modeling climate change prospects are still significant, especially on regional and local scale, where currently mainly climate change tendencies are available. Furthermore, restrictions for the assessment and management of climate change challenges to seaports exist, like differences in planning horizons, lack of relevant information and treatment of uncertainty, limited climate change awareness amongst stakeholders, financial restrictions, and complexity of the decision making process with respect to climate change issues. General statements on the vulnerability of seaports to climate change are not possible. In some cases, already today climate change related problems in seaports are obvious, like changing properties of permafrost soils and related foundation problems in high latitudes. Other climate change impacts will influence port planning and operation only at a later stage, because in short and mid-term only minor effects are effective or the vulnerability of the system ‘seaport asset’ to the respective climate trend is only marginal. Additionally, ports located at the open sea have to bear

other loads than estuary or lock-separated ports and

the sensitivity of cargo handling with respect to wind

and wave loads differs with cargo type.

The climate vulnerability of a seaport is defined by

its exposure, sensitivity and adaptation capacity to cli mate change. The determination of the exposure of a specific port has to be based on an analysis of climate change impact scenarios on a regional scale. Locally relevant climate change scenarios have to be defined on the basis of existing global and regional findings. Considering climate change aspects in port planning always means the inclusion of uncertainties of climate change projections as well as changes in predictions, which will have to be accepted not only today but also in future. All planning must be done on the basis of scenarios including each a range of values of spe cific climate data like temperature, sea level rise, or rainfall intensity. The identification of current vulner abilities like port downtime and/ or damages because of e.g. fog, extreme water levels or storms provides valuable indications to assess potential future key problems.

Adaptation measures in seaports with respect to cli mate change can be distinguished in active and reactive measures.Active measures are the appropriate solution for climate change sensitive port assets in case their expected lifetimes are long and a later adaptation to climate change impacts may be connected with a sig nificant increase in costs. Examples of active measures are:

- Using heat resistant materials like concrete pave ments instead of asphalt
- Using larger units in breakwater construction due to an anticipated increase in wave loads with respect to sea level rise and/or larger wind speeds
- Provision of enough space and strength for refitting

Special session: Innovative solutions for adaptation of European hydropower systems in view of climate and market changes

Figure 1. Worldwide development of hydroelectric

pumped-storage installed capacity by regions (source: EIA).

drop is effectively used by energy-storage facilities.

The recent increase of available renewable energies

is therefore seen as a potential driver for additional

storage necessities such as pumped-storage systems,

which contribute to balance the grid by reallocating

different sources of energy supply. Germany intends

to provide 80% of its energy from renewable sources

by 2050 (German Advisory Council on the Environ

ment, 2011), which will require a corresponding effort

in terms of added storage capacity. According to the U.S. Energy Information Admin

istration (EIA), as of 2012, pumped-storage systems

accounted for approximately 132 GW worldwide. Fig

ure 1 shows the evolution of pumped-storage installed

capacity according to its geographical region. Asia

and Oceania has the highest pumped-storage capac

ity, followed by Europe and North America, whereas

the rest of the world accounts for less than 6GW.

The leading countries in pumped-storage are: Japan

(27 GW), United States (22GW) and China (21G W).

Germany occupies the 6th position, after Italy and

France, with 7 GW of installed capacity which rep resents approximately 4% of its total installed electric capacity.

1.3 Underground pumped-storage

Although pumped-storage is seen as a reliable tech nology to store high amounts of energy, its varia tion into underground pumped-storage has not been equally explored and many challenges still remain for its full implementation. The underground pumped storage involves not only geotechnical uncertainty but also construction risks and operational constraints sig nificantly larger compared to conventional pumped storage schemes. The underground pumped-storage consists of hav ing at least one of the two reservoirs built underground. This option has been proposed in particular in the pres ence of former mines, for which the existing infras tructure seems an attractive available resource that can significantly reduce construction costs. Some exam ples of feasibility studies carried out for underground pumped-storage are: Summit project, in Norton Ohio; Maysville in Kentucky; Mount Hope in New Jersey; Elmhurst Quarry in Illinois; Riverbank Wiscasset in Maine To this day, however, there is no underground pumped-storage in operation or in construction and the technical and economic feasibility of carrying out such projects remains uncertain. An initiative from a group of five partners in Germany (University of DuisburgEssen UDE, Ruhr University of Bochum, Rhine Ruhr Institute for Social Research and Political Consultancy RISP, former RuhrkohleAktiengesellschaft now RAG, and former Deutsche Montan Technologie DMT) supported by funds of the European

Union, analyzed the feasibility of using some of the current coal mining facilities in the Ruhr region as lower reservoirs for a PSH project. This study summarizes some of the main findings of this feasibility study, considering hydraulic, geotechnical, constructability, geological, and economic constraints. The research focused on the analysis of Prosper-Haniel mine, as this infrastructure will soon become available and it is foreseen as a reliable option to increase energy storage options in Germany as part of its energy goals for 2050. 2 INTEGRATED ASSESSMENT 2.1 Description of Prosper-Haniel mine Prosper-Haniel mine is located near the city of Bottrop in the Ruhr region in Germany. It started to operate during the 1850s and will be shut down in 2018. The mine was excavated in 7 well identified levels of tunnels ranging from 350 m to 1320 m depth, which can be reached from 5 different shafts (Prosper IV, Prosper V, Lohberg Hünxe, Franz-Haniel 1 and Franz-Haniel 2) and 1 inclined tunnel.The latter has a slope of 22% and provides access only to levels 5 and 6. Each of these levels was built in a chronological order in which the levels located closer to the surface were first developed and new and deeper levels were excavated once the previous levels were nearly fully exploited. The mining facilities are usually developed as temporal infrastructure only intended to endure a short period of time, enough for the process of coal extraction. Despite the highly dynamic construction environment of these infrastructures, there is a basic network of tunnels that are preserved as main conduits to transport and extract the coal and as communication lines. This network offers more stable geotechnical conditions and a more permanent environment for long term hydraulic infrastructure. In general, the slope of these tunnels and their geometrical design are favorable to store large volumes of water. Figure 2 shows a schematic description of the main networks of levels 2 to 6 and their available access routes (level 7 offers very limited storage volume and level 1 has scarce information). A more comprehensive description of this mine is provided by Alvarado et al (2013). The development of pumped-storage using the underground shafts and tunnels of coal mines in the Ruhr region is appealing due to the lack of favorable Figure 2. Scheme of current available shafts and provided access to each levels tunnel network in Prosper-Haniel mine. topographical conditions in this area. The almost flat topography offers no real potential for conventional PSH. However, sufficient water head could be provided by the existing shafts in the mine, while the tunnel networks could offer sufficient volume to be used as the lower storage. The feasibility study analyzes which parts of the mine were more suitable to develop an underground PSH and/or extend the system with complementary infrastructure. We analyze this system under two potential conditions: the first was a closed-loop system, which consists of having the pumped-storage project isolated from a naturally-flowing water source. This contrasts to the open-loop systems which consider a continuous connection between the system and the surrounding flowing water sources. The open-loop system was particularly relevant to analyze the environmental impact of developing an underground pumped-storage project in the region. However, the closed-system proved to be a more robust concept. 2.2 Capacity for energy storage A first glimpse to the potential capacity of the system can be obtained by computing the energy produced by the combined characteristics of available storage and water head from: where η is efficiency, ρ is density of water in kg/m 3 , g is gravity in m/s 2 , H is water head in m and V is storage volume in m 3 . Figure 3 shows the potential energy at each level of Prosper-Haniel mine, compared to other hydroelectric projects in the region. Based on the previous computations, two options for level 6 were suggested: one which would only consider currently open sections of the tunnel network Figure 3. Storage capacity for each of the levels in Prosper-Haniel mine, compared to existing projects in the region. (L6a) and one that would include additional sections of tunnels which were currently closed (L6b). Levels 2 to 5 showed a very low potential in terms of total energy per cycle are were therefore discarded for further assessments. Despite the great potential of energy storage in level 6, future ground water levels in the mine are likely to compromise its use. It is expected that after the closure of the mine, the underground water level will be upraised from approximately 1200 mbsl (meters below sea level) at present to 580 mbsl. This would clearly compromise the potential use of infrastructure below the new ground water level and a submerged lower reservoir would have additional challenges in terms of geotechnical stability. An alternative option as underground storage was therefore recommended, which considered the construction of a new level above the future ground water level. This option consists on the excavation of a new tunnel at approximately 530 mbsl which still makes use of the existing shafts and infrastructure. This is particular important since the cost of building a shaft is significantly higher than the costs of excavating tunnels. The extension of this new tunnel depends on the economic feasibility of the project. The project has to be able to participate in the secondary power market (Daou Pulido, 2015), which forces the facility to be able to provide energy for at least four consecutive hours. At a net discharge of 40 m^3/s, this means having a storage capacity of 600,000 m^3 equivalent to a total length of approximately 15.5 km. The geometry of the new tunnel was defined as a close ring with a slope of 0.2% and an internal diameter of 7m, based on hydraulic conditions explained later in this study. This new ring structure offers approximately 820 MWh of energy per cycle (depicted in Fig. 3).

Figure 4. Schematic configuration of main components around the chambers of the underground pumped storage concept.

2.3 Conceptual technical design of components

The most relevant technical aspects are related to: i) the cavern for the electromechanical machinery and ii) the storage structure for the lower reservoir. Due to spe cific changing geological conditions at the location of the existing shafts, a sand-stone layer was identi fied as the most robust rock structure to host the main caverns. The roof of the cavern therefore would be completely excavated within this layer. The geotech nical analysis determined that the caverns could have maximum dimensions of 60m length, 20m width and 25m height. Figure 4 shows a schematic view of the main components surrounding the main chamber. A main cavern hosts the reversible pump-turbines and generators, whereas a complementary cavern hosts the transformers and other electrical components. Due to the restrictions in the cavern dimensions the selection and number of turbines had to be limited. Such con straints limited the equipment to two reversible Francis machines with horizontal axis. The selection of two machines provided a good trade-off between available space and flexibility for power generation. The latter is crucial to the economic feasibility of the project as it allows the facility to increase its participation in the energy markets and cover a wider range of energy demands. The penstock of the system would be located inside the existing shaft Franz-Haniel 1, embedded in filling concrete, and the nearby shaft Franz-Haniel 2 would be used for the electric lines to connect the genera tors with the grid. This would be done by using Gas Insulated Lines (GIL) since a conventional cable con nection would not be able to be installed vertically over such a long distance, due to its own weight. From a hydraulic perspective, an initial look into the slope of the tunnel networks in Prosper-Haniel mine showed which levels had a favorable draining condi tion towards the shafts. Level 5 has a slope draining in opposite to the shaft and was therefore discarded as an option. Level 4 has a similar condition with the added problem that it has long been closed and additional costs would be required to reopen the main tunnels. Level 3 has a very limited storage and its net work suffered the same difficulty in terms of drainage.

Figure 5. Range of uniform flow regimes for different

slopes along the tunnel network and corresponding discharge capacity. The only two levels that proved to have a favorable condition were level 2 and level 6. As explained before, level 6 had to be discarded due to future water levels, whereas level 2 would only be able to produce approximately 90 MWh and it was foreseen as a very expensive option. For a more detailed explanation see Alvarado et al., 2013. The alternative of excavating a new ring structure at 530mbsl presented additional challenges in terms of construction. In order to minimize costs, the tunnels would need to be excavated with a tunnel boring machine, which would need to be installed at approximately the same depth of 530 mbsl. This requires the excavation of a chamber with similar dimensions as the main chamber. Additionally, a supplementary network of tunnels of at least 0.5 km is required in order to connect all the components of the system. Under the concept of building a new tunnel, the limiting factor for the cross section becomes its geotechnical stability (from a hydraulic point of view a larger cross section becomes a more efficient solution). Therefore, the internal diameter was limited to 7 m. The slope is then determined by the fastest drainage response of the system, while keeping the network under subcritical regimes (Froude number was limited to 0.80) in uniform flow conditions. Figure 5 shows the critical depth for a section of 7 m with low roughness (0.015), as the most extreme scenario in the system. From this figure it is clear that a slope of 0.2% satisfies the previous conditions and was therefore suggested for the new ring. 2.4 Non-technical assessment of the project In addition to the technical feasibility, the social and political acceptance of an underground pumped storage plant was studied. This was an essential prerequisite for the potential development of the project. It was based on survey studies which assessed the acceptance of a future underground pumped-storage project in the region. The results showed that energy transition towards renewables sources in the Ruhr region has a high degree of consent (Grunow et al, 2013). The population also agrees on the construction of new plants of renewable energies to support the energy transition within their own communities, while only 2.7% rejected both the energy transition and the concept of energy storage through pumped-storage plants (Grunow et al, 2013). The ecologic effects were assessed under the assumption of having an open-loop system. Results showed that the released discharges into the nearby water bodies would have a positive effect in terms of salt concentrations due to currently high concentrations in the river network. The economic feasibility of the project was analyzed taking into account two energy markets in Germany.

These markets are the Spot Market (for energy trading) and the secondary control power market (providing electricity balancing services). Due to current, as well as expected future conditions in the energy market, the electricity storage cost for implementing this project would range from 30 to 50 Euros/kWh. This resulted in negative net present values which compromise the feasibility of the project. However, uncertainty in the development of the energy markets, as well as in the assessment of construction and operational costs, was recognized to introduce significant uncertainty in the economic assessment (Daou Pulido, 2015). 3 HYDRAULICS 3.1 Water supply As explained in 2.1, the system was essentially analyzed as a closed-loop project. This means that the system needs to be initially filled in from an off-site water source. Three options were analyzed as alternatives for this: i) use drinking water supply at the site, ii) build a surface connection between the Rhein-Herne canal (shipping channel) to the upper reservoir, iii) build a surface connection between the Rhein-Herne canal and the inclined tunnel (see alternative routes shown in Figure 6), which will then be used to fill in the system. The inclined tunnel connects levels 5 and 6 of the mine and it would be the main access route to build the new ring structure, as well as to operate and maintain the system. After an economic assessment, option (ii) was discarded as the most expensive one (also the most invasive in terms of construction). Option (i) costs approximately 750.000 Euros for a single cycle of filling the system, similar to option (iii). The latter, however, offers a very low marginal cost for added filling cycles and it is therefore the most robust option for the system. Figure 6 shows a diagram of two possible connection routes between the channel and the entrance to the inclined tunnel in Bottrop. 3.2 Hydraulic design The focus of the hydraulic design was placed on the lower reservoir, as it is the main driver for the rest of the Figure 6. Overview of connection options from the port of Bottrop on the Rhein-Herne canal to Prosper-Haniel mine. components of the project. A one-dimensional model was implemented in order to analyze hydraulic losses along the system and provide efficiency curves for the future operation of the reservoir (Alvarado-Montero et al, 2013). Moreover, the model allowed analyzing dissipation waves and the hydraulic behaviour of the system under different operation scenarios. The simulations were carried out with a roughness factor of n Manning of 0.015 which represents a smooth condition for the reinforced concrete of the tunnel structure. Such condition is considered as the most critical one since it dissipates less energy and produces higher values for transient flows impacts. Two sets of experiments were carried out using this model. The first consisted on the complete filling and emptying cycles of the system. The emptying process of the lower reservoir was simulated until reaching a minimum value of –512 m as illustrated in Figure 7. It is important to notice that the total pressure at the entrance (node 1) reduces to almost –522 m when bringing the reservoir to its minimum elevation of –512 m using this discharge. Such condition is particularly critical since it would produce cavitation problems at the machine. Therefore it was recommended to reduce the operation of the system to only one unit during the last stage of the emptying process. This means having a reduced pumping discharge of 20 m 3 /s below –509 m, for approximately 15 minutes until reaching the minimum level of –512 m. The water level at the turbine reaches a minimum value of –515, leaving approximately 3 m of gross water head under critical operation conditions. The system takes about 0.5 hour to reach back a rest level condition. On the other hand, the filling process showed a smooth transition during the complete simulation of 4 hours under maximum capacity and didn’t require any special restriction on the operation. Our second set of experiments simulates typical turbining-pumping cycles, in order to analyze possible resonance effect of propagation waves along the

Figure 7. Emptying process of ring structure, without oper

ating restriction, and with operating restriction (noticed at

proximity to the turbine, at Node 1).

system (Figure 8). A first case was simulated using

4 production cycles (turbine mode) of 5 min alter

natively switch on after 5 min of pause. It produces

shockwaves of approximately 5m that are nevertheless

dissipated fast enough between each cycle. It is also

interesting to notice that the closure of each production

cycle produces a negative reaction of pressure along

the system with a similar dissipation time. A second

case was simulated using cycles of 5 min of turbining alternating with pumping mode, both on full capacity using 40 m^3/s. Despite the sudden change on the direction of the flow, the behaviour in terms of pressure is very similar to the previous simulation, in which shockwaves have a range of ±5 m with dissipation times of less than 5 min.

3.3 Special structures

One of the special structures for the pumped-storage system is the inlet/outlet structure of the lower reservoir which combines the outlet of the two francis pump-turbines to the storage network (see Figure 4). Its main purpose is to ensure favorable flow conditions for turbine and pumping modes of the storage power plant and reduce hydraulic losses that could compromise the overall efficiency of the system under different scenarios. The optimization of the inlet structure was done by analyzing different physical setups using a three-dimensional flow simulation in the open source software OpenFOAM. The numerical setup includes an unscaled grid with the size of 294,000 hexahedron cells which varies between lengths of 2 to 50 cm. To solve the Reynolds-Averaged-Navier-Stokes equations the PIMPLE-algorithm (merged PISO-SIMPLE) was used. In terms of turbulence modeling the k-e turbulence-model was implemented. The simulation examined the local hydraulic pro cesses in particular the flow velocity field and pressure

conditions as well as the turbulence effects for flow

Daou Pulido, A. 2015. Wirtschaftliche Bewertung eines Untertage-Pumpspeicherwerk-Konzepts unter der Bedingung technischer und ökonomischer Unwägbarkeiten, Energie und Nachhaltigkeit, Band 21, Ed. LIT, Bochum

Dymek, T. Energiewirtschaftliche Rahmenbedingungen des Betriebs von Pumpspeicherkraftwerken. In: Korrespondenz Wasserwirtschaft 6 (2013), Nr. 5, S. 270–275.

Fischer, P., 2011. Erneuerbare Energien dank Bergbauressourcen. In: Steinkohle 2011 – Energie für neue Wege; Gesamtverband Steinkohle e. V., VGE Verlag GmbH, ISSN 0343-7981, S. 52–56, Essen, 2011

German Advisory Council on the Environment (SRU). 2011. Pathways towards a 100% renewable electricity system. Ed. Erich-Schmidt GmbH, Berlin.

Geschäftsführung des Gesamtverbands Steinkohle e.V. (GSVT), 2012. German Coal Association Preparing to phase out.

Knight Piésold Consulting, 2010. BC Hydro Evaluation of Pumped Storage Hydroelectric Potential. Screening Assessment Report (Ref. No. VA103-313/1-1).

Manwaring M., Mursch D., Tilford K., 2012. Challenges and Opportunities for New Pumped Storage Development – A white paper developed by NHA’s Pumped Storage Development Council. Niemann, A.; Perau, E.; Schreiber K.; Koch, H.-J. (2014): Chancen und Risiken untertägiger Pumpspeicherwerke in Steinkohlebergwerken im Ruhrrevier, Zeitschrift Wasserwirtschaft 1/2 2014, S. 66–69; ISSN: 0043-0978 REN21, 2013. Renewables Global Futures Report – Renewable Energy Policy Network for the 21st Century and Institute for Sustainable Energy Policies. Steffen B., 2011. Prospects for pumped-hydro storage in Germany. Chair for Management Science and Energy Economics, University of Duisburg-Essen. Uddin N., 2003. Preliminary design of an underground reservoir for pumped storage. Geotechnical and Geological Engineering 21; Kluwer Academic Publishers. Yang C-J., Jackson R.B., 2010. Opportunities and barriers to pumped-hydro energy storage in the United States. Renewable and Sustainable Energy Reviews. Yang C-J., 2010. Pumped Hydroelectric Storage. Duke University, Durham, North Carolina, USA. U.S. Energy Information Administration:

International Energy Statistics. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4
Feasibility assessment of micro-hydropower for energy recovery in the water supply network of the city of Fribourg I. Samora Laboratory of Hydraulic Constructions, École Polytechnique Fédérale de Lausanne, Lausanne, Switzerland Civil Engineering Research and Innovation for Sustainability, Instituto Superior Técnico, Lisbon, Portugal P. Manso, M.J. Franca & A.J. Schleiss Laboratory of Hydraulic Constructions, École Polytechnique Fédérale de Lausanne, Lausanne, Switzerland H.M. Ramos Civil Engineering Research and Innovation for Sustainability, Instituto Superior Técnico, Lisbon, Portugal ABSTRACT: In water supply systems there is potential for hydropower production at several scales, from small hydro to micro. The energy recovery within the urban water supply networks is a type of micro-hydro that may be useful for the control of excessive pressures. On the other hand, local decentralized production of electricity may have multiple uses, considering self-consumption at local grid level or storage. However, there is still a lack of technologies and specific solutions for such applications, since the flows are highly variable and the available heads are small and limited to service pressures. A scheme specially conceived for water supply networks making use of a micro turbine is proposed.Adequate positioning conditions are identified using a search algorithm which considers both the assessment of the energy production and sizing of the main equipment and works. Typical schemes are proposed where the installation of up to four turbines is possible within the same buried chamber created around an existing pipe. Preliminary results obtained for a network case study show that the implementation of the proposed energy recovery solution is feasible. The installation of a by-pass revealed to have a key role in the feasibility of each solution, demanding customized engineering judgment. Further testing of the search algorithm as well as a first in-situ implementation of the scheme may be foreseen in the near future. 1 INTRODUCTION The present concerns with environment and energy efficiency contribute for a growing new interest on small and even micro-hydropower. These small scale technologies, micro-hydro designating productions below 100 kW of installed power (Ramos et al. 2009), allow a decentralized supply for local demand, which has the advantage of reducing transmission losses (Weijermars et al. 2012). One of the most promising applications of microhydropower lies within water supply systems (WSS). These are systems which are pressurized and where the pressure control is important to avoid water

losses and pipe damage (Carravetta et al. 2012, Xu et al. 2014). The use of turbines instead of pressure reduction valves (PRV) for the excess pressure dissipation allows to recover part of this energy (Ramos et al. 2010, McNabola et al. 2014, Su and Karney 2015). Also, the implementation in existing water supply infrastructure has the advantage of smaller costs (Sitzenfrei and von Leon J 2014). Although many studies exist in literature on pressure control in urban water systems, there is still a lack of technologies and specific solutions for energy recovery in these conditions. Carravetta et al. 2012 proposed the use of pumps as turbines (PAT) between district metered areas and Corcoran et al. 2015 studied the use of PAT, Francis and Kaplan turbines. In the present work, a feasibility study is carried for the installation of micro-turbines within the urban areas of the WSSs. An arrangement is defined, with inline tubular propellers installed within existing pipes of the network, and the main quantities are estimated in order to have an economic analysis. The ideal location of such an arrangements in a water supply network depends on numerous factors: the flow rates, which in a urban system are highly variable along the day; the available head, since the micro-hydro operation should not affect the quality of the service to the population and thus minimum service pressure ought to be guaranteed; and the geometry of the network, due to the distribution of the flow within closed meshes of a network. To identify the optimal placement of the turbines, an algorithm was developed to optimize the economic value. This algorithm, coupled with a hydraulic solver, is an upgraded version of a previously developed work

Figure 1. Model of the runner of the 5BTP, with 85 mm of

diameter.

Figure 2. Schematic lay-out of a chamber equipped with

four turbines.

based on the Simulated Annealing strategy (Samora

et al. 2016). The city of Fribourg, Switzerland, was used as a case

study to obtain estimations on the energy produced

with the proposed arrangement and of its economic

value.

2 MICRO-HYDROPOWER SCHEME

The five blade tubular propeller (5BTP) is appropri

ate for energy recovery in water supply systems since

its operation is based on variable flow rates and low

heads. This turbine was first developed at Instituto

Superior Técnico, Portugal and recently it has been

subject to experimental tests in Switzerland, in a coop

eration between the École Polytechnique Fédérale de

Lausanne and the University of Applied Sciences and

Arts of Western Switzerland. The turbine, as shown in Figure 1, consists of a run

ner with five fixed blades attached to a bulb upstream

and to an axis downstream. The axis connects to an

external generator, leaving the pipe through a 45 $\circ$

curve. For the placement of this turbine within a water

supply system, the configuration shown in Figure 2

is proposed. This scheme is based on the construction

of a buried concrete chamber around an existing pipe

where the installation of up to four turbines is possible. The diameter of the runner D_t is always inferior to

the diameter of the existing pipe D_p . The electromechanical groups are composed by

the turbine, the generator and a frequency converter that controls the rotational speed according to flow measurements. For construction and maintenance, access to the equipment is required and therefore the isolation of the chamber from the network is an aspect to take into account. As most WSN are composed of meshes, there is often redundancy in the supply and valves are in place to isolate branches. Nevertheless, if we consider a node fed with no redundancy, a bypass must be created and the hydraulic circuit adapted. The creation of such bypass needs the installation of three maintenance valves, that can be placed in D_t sections. 3 METHODOLOGY AND CASE STUDY 3.1 Search algorithm Water supply networks are complex systems, with variable flows, redundancy of supply and pressure restrictions. Hence, the decision of the placement of the turbines within a network needs a deep analysis of all factors involved. To perform this analysis, a search algorithm which is based on previous works (Samora et al. 2016) was used. In this algorithm, a simulated annealing process was developed to locate the placement of turbines in a network that maximize the energy production. In this work, the algorithm was modified to allow up to four turbines in the same branch of the network and the cost function was changed to: where $X = (x_1, \ldots, x_n)$ is the solution vector, representing the placement of N_t turbines, in this particular case $N_t = 4$, and $NPV_{20years}$ is the net present value resulting from the sum of the cash-flows over 20 year of operation. Only investment costs (IC) were considered and it was assumed they would take place in the year prior to commissioning. Hence, the net present value after 20 years of operation is given by the following discounted cash flow model: where E_{annual} is the annual energy production with the solution X, t is the selling energy tariff and r is the discount rate.The annual energy production is given by where g is gravitational acceleration (m/s^2), ρ is the water density (kg/m^3), T corresponds to the time window of one year with an hourly time step �t and, for each n turbine, Q_n (m^3/s) is the flow discharge, H_n (m) Figure 3. Network of the water supply system of Fribourg, Switzerland, used as case study. is the net head in the turbine and η_n is the total efficiency. The total efficiency is given by the turbine and generator efficiencies. The first depends on the efficiency curve of the turbine, which wasstudied and the second was considered constant and equal to 85%. 3.2 Case study The search algorithm was applied to the drinking water supply network of the city of Fribourg, Switzerland. The model, with 2972 links, 2805 nodes, 15 PRV and 7 water tanks was provided by the Industrial Services of Fribourg (Figure 3).The network has a total of 216 m of elevation difference and 130 l/s of average daily consumption over 24 h. A minimum pressure restriction of 30 m was assumed, taking into account the typical height of buildings in Switzerland and the need to supply with enough pressure to the consumers. Based on real data and on the model implemented by the network managers, an average consumption of 0.108 l/s was associated to each demanding node and a typical pattern of hourly variation along the day was applied. To estimate the revenues, the electricity sell price adopted was the current feed-in-tariff in Switzerland of 0.33 CHF/kWh for this type of power plant (SFC 1998, SFOE 2015). Discount rates of 4%, 6% and 8% were considered in this study. Finally, to estimate the investment costs, the main quantities of the power plant presented in Figure 2 were estimated in relation to the pipe and turbine diameters and installed power. The elements considered were: stainless steel, concrete, excavation, earth fill, electromechanical equipment, isolation valves, flowmeters. Unit prices, presented in Table 1 were associated to these quantities. The installation of isolation valves is dependent on the existence of redundancy to the nodes of the target branch. However, since the need for a bypass is restricted to short periods of time, it was considered that the minimum pressure during construction and maintenance is 15 m.

Table 1. Unit prices.

Element	Unit price
Stainless steel	7 CHF/kg
Reinforced concrete	250 CHF/m^3
Excavation	30 CHF/m^3
Earth fill	20 CHF/m^3
Electromechanical equipment	1 CHF/W
Maintenance valve w/ wheel drive	190 000 CHF/m^2
Flowmeter	550 CHF/unit

Table 2. Results for the installation of four turbines. NPV 20years (kCHF)

X	E (MWh/year)	r: 4%	6%	8%
(2730, 2730)	121	513	428	362
(2730, 2730, 2730)	128	546	456	386
(2091, 2730, 2730, 2730)	144	573	473	396

4 RESULTS

The search algorithm was run for the installation of two, three and four turbines in the case study network. The produced energy and the net present value after 20 years of are presented in Table 2 for the obtained solutions. Pipe 2730 (identified in Figure 3) has a clear superior potential for hydropower production. In Table 2, all the solutions for the position of turbines imply the use the same pipe except for four turbines. A turbine imposes a head loss that constrains the passage of flow and, even though the solution with four turbines implies the construction of two chambers, the effect of the limitation of flow discharge with four turbines in pipe 2730 does not seem to compensate. No particular limit to this reduction was imposed in this case, as it was assumed constant levels in all reservoirs and tanks and continuous consumption in the nodes. To illustrate the comparison between costs and energy production, in Figure 4 are presented the break down of costs for these three solutions and also of the solution where all four turbines placed are in pipe 2730. The diameter of the runner in each solution is dependent on the maximum flow in the pipe, and so it is not the same in all cases. It can be seen from Figure 4 that no maintenance valves are present in the breakdown of investment costs

for any of the solutions. Therefore, these position in

the network have redundancy in the supply, not requir

ing the constructions of a by-pass. In fact, pipe 2730 is

located in one of the main paths connecting an external

tank to the city. This path is the region of the network

with the highest flow rates and the pressures are high.

Also, a PRV is already in service in this path. Never

theless, despite the importance of this path, the city

Banki-Michell micro-turbines for energy production in water

distribution networks

V. Sammartano & P. Filianoti

Department of Civil, Energy, Environmental and Materials
Engineering, Università Mediterranea di Reggio

Calabria, Reggio Calabria, Italy

G. Morreale

WECONS, Palermo, Italy

M. Sinagra & T. Tucciarelli

Department of Civil, Environmental, Aerospace, Material
Engineering, Università degli Studi di Palermo,

Palermo, Italy

ABSTRACT: The design of a new Banki-Michell type turbine is
proposed for electricity production from

pressurized pipes. The advantage of the new turbine is its
simplicity, as well as the direction of the outlet flux,

which remains in the same plane of the impeller and of the
original pipe axis. The turbine has a total length of

10-12 times the diameter of the hosting pipe. The design

procedure is validated with a given set of input data using ANSYS CFX numerical solver: the specific energy difference between the inlet and outlet section, the expected discharge, the impeller rotational velocity. Solution is found for a given set of discharges after changing the impeller rotational velocity in order to keep the relative velocity always equal to 2. Efficiencies remain above 84.3% for all the tested discharges, with a maximum/minimum discharge ratio equal to 4.5 and a peak value of 86.8%.

1 INTRODUCTION

Banki-Michell turbines combine simplicity with efficiency and represent a possible alternative to the use of PATs (Pump As Turbines) for hydropower energy production at the end of aqueducts delivering water to urban tanks (Chattha et al. 2010; De Andrade et al. 2011; Khurana & Kumar 2011; Sinagra et al. 2014). The traditional outlet of this type of turbine is at atmospheric pressure and this precludes its allocation along aqueducts or insideWater Distribution Networks (WDNs). The use of in-line turbines could allow the selection of these sites, where a pressure reduction would not affect discharge regulation, but rather limit the pressure downstream of the turbine and reduce water losses. In the following, starting from the design criteria previously developed for Banki-Michell turbines

(Sammartano et al. 2013, 2015), a new Banki-Michell turbine aimed to overcome this limitation is proposed. Moreover, a detailed analysis is carried out to investi gate the variation of both the efficiency and the total head dissipation along with the change of the impeller rotational velocity, assuming the electric regulation of the turbine to be available.

2 TURBINE DESCRIPTION

The Banki-Michell turbine has a constant section along any plane normal to the impeller axis. It is composed by four main parts (Fig. 1): the convergent pipe, the nozzle, the rotating impeller and the diffuser. The convergent pipe is aimed to accelerate the particles, transforming most of the potential pressure energy in kinetic energy and the nozzle works as distributor of the discharge entering the impeller through the inlet surface. The impeller inlet and outlet surfaces are part of a cylinder, with generator lines parallel to the axis and are laterally bounded by the two impeller disks. The two impeller disks form a single solid with the blades that have semi-circular shape and constant inner radius. The water flow crosses twice the blade channels, before leaving the impeller to enter the diffuser. This part, which is missing in the original Banki-Michell turbine for zero-pressure outlet flow, is designed in order to minimize the dissipation of the particle specific energy along the path between the impeller and the outlet section of the turbine case. The turbine is assumed to replace the aqueduct pipe for a short length, of the order of 10–12 diameters and to have the inlet and the outlet sections equal to the section of the aqueduct pipe, aligned on the same axis. The main hypothesis adopted for the diffuser design are: 1) the impeller outlet tangential velocity is zero; 2) the impeller outlet radial velocity is constant along all the impeller outlet surface; 3) given a straight line passing through the trace of the axis, the mean velocity component of the particles inside the diffuser, on the direction normal to the straight line, is constant (Fig. 2). Conditions 1) and 2) are perceived through the Figure 1. Scheme of the Banki-Michell turbine. Figure 2. Scheme of the turbine nozzle, impeller and diffuser. optimal design outlined in Sammartano et al. (2015). Condition 3) can be obtained from the mass conservation and the application of hypothesis 2), if a proper profile AB is given to the external diffuser surface (Fig. 2). After point B, a small restriction of the section normal to the mean velocity (see line CD in Figure 2) is applied in order to avoid the generation of vortices occurring because of the change of the path curvature. After section BD there is first a curve with constant rectangular section and then a divergent pipe which allows the transition from the upstream rectangular cross-section to the downstream circular one. The main advantage of the proposed turbine is the low construction cost for given nominal power. Another important advantage is that the direction of the outlet flow is on the plane of the rotating impeller and this strongly simplifies the return of the flow to the original pipe direction. For this reason the turbine is particularly suitable for in-line installation in existing pipelines. If properly coupled with electric and/or hydraulic regulation systems, the device could also be an alternative to the classical PRV energy dissipation devices in pipe networks, with the extra benefit of producing even a limited amount of electric energy. The design of the water turbine is supported by CFD simulations that allow to optimize the geometry of the turbine and also to evaluate the performance of the machine according to different hydraulic conditions, such as a large range of specific energy drops and water discharges (Aziz and Desai 1993, De Andrade et al. 2011; Haurissa et al. 2012; Sammartano et al. 2013; Sinagra et al. 2014). 3 DESIGN PROCEDURE Numerical and Experimental investigations (Sammartano et al. 2015; Sinagra et al. 2015) showed that in a Banki-Michell turbine the maximum efficiency is obtained when the absolute tangential velocity ($V_t = V \cdot \cos\alpha$) is about twice the velocity of the reference system ($U = \omega \cdot R_1$), such that: where V_r is the velocity ratio, V is the impeller inlet velocity, α is the attack angle, ω is the rotational velocity of the impeller and R_1 is the outer radius of the impeller (Fig. 2). Assuming that the velocity V is proportional to the root of the specific energy at the impeller inlet, E, by means of a velocity coefficient C_V that accounts for the particle energy losses, we get: Unlike the traditional Banki-Michell turbine, in this device the water gauge pressure along the impeller inlet and inside the impeller is greater than zero, and the optimal geometry strongly depends on the occurring velocity field. Starting from previous studies of Sammartano et al. (2013, 2015) the authors obtained an empirical expression for the specific energy E using a series of CFD simulations, taking into account different hydraulic conditions (specific energy drop and water discharge). The numerical study led to the following approximating formula: where ΔH is the specific energy drop between the inlet (H_m) and the outlet sections of the turbine (H_v) and the second term is the water gauge pressure around the outer boundary of the impeller, which mainly depends on the rotational velocity and on the outer radius of the same impeller. The coefficient ξ was estimated on the basis of a series of CFD analysis solving several tests with different total energy drops ΔH and a wide range of water discharges Q. In Table 1 the input data of the numerical simulations and the values of the coefficient ξ computed from Eq. (3) and the solved energy drop ΔH are shown. The tests showed that the coefficient falls in the range between 2.258 and 3.010. Because the kinetic energy associated to the second term of the r.h.s. of Eq. (3) is usually small with respect to the first one, the average value of the computed ξ can be used in practice for the turbine design. The outer radius of

Table 1. Coefficient ξ .

D_1	D_2	N_p	H_m	H_V	Q	ΔH	ξ
mm	mm	–	m	m	m^3	m	–
154	92	43	50	30.3	0.120	19.70	2.746
163	114	40	60	30.1	0.200	29.90	2.876
202	141	43	60	30.1	0.200	29.93	2.690
189	132	39	70	30.3	0.150	39.71	2.900
233	163	48	40	20.3	0.150	19.70	2.671
233	163	50	40	20.1	0.150	19.90	2.729
233	163	52	40	20.0	0.150	19.97	2.673
233	163	45	40	20.2	0.150	19.76	2.652
234	164	46	40	20.3	0.150	19.73	3.010
134	101	40	35	25.7	0.030	9.35	2.656
123	98	35	35	19.0	0.040	16.00	2.555
125	88	45	35	34.0	0.010	1.03	2.636
127	114	30	35	14.3	0.045	20.70	2.934

138 90 50 35 30.9 0.015 4.13 2.258

the turbine impeller can be obtained from Equations 1–3, given the upstream (H_m) and downstream (H_v) specific energy, as well as the discharge. The blades geometry is defined investigating by means of CFD analysis different numbers of blades N_b and diameters ratio D_2/D_1. The shape of the diffuser is aimed to keep almost constant the velocity norm, in order to avoid energy losses due to particle decelerations. The section of the diffuser, along any plane embedding the turbine axis, has a rectangular shape. The diffuser is bounded by two lateral planar walls and two curved walls, a bottom (AB) and a top (CD) one (Fig. 2). This particular shape is aimed to convey the radial trajectory of the particles leaving the impeller into a common direction. This is accomplished by keep ing constant the mean velocity computed through any planar section embedding the turbine axis. Due to an optimal impeller design, the tangent component of the particle velocity at the impeller outlet is almost zero and the outlet velocity norm is almost equal to the radial component of the inlet velocity (Fig. 2). The bottom wall (AB in Figure 2) has a cylindrical shape, and the generating straight line has a distance from the runner’s axis increasing with its angular coor dinate (measured with respect to the nozzle tip). The distance is set by assuming the velocity of the parti cles located on the plane shared by the turbine axis and the generating line to be normal to the same plane and with a norm almost equal to the radial component of the particle velocity at the impeller outlet. The width B of the impeller was estimated according to the continuity equation applied at the inlet of the impeller:

where Q = design flow rate; and λ= impeller inlet angle (Fig. 2). 4 FLUID DYNAMIC INVESTIGATION In order to verify the design procedure and to optimize the turbine geometry a series of numerical simulations were carried out using the Ansys CFX commercial code. The CFD simulations were performed solving the Reynolds-averaged Navier Stokes (RANS) equations, coupled to the SST turbulence model. This model applies the k-ε model in the free shear flow and the k-ω model (Wilcox 1993) in the inner region of the boundary layer (Vieser et al. 2003; Menter et al. 2003a). The SST model has been used in most of the turbomachinery applications found in the literature. Menter et al. (2004) applied the SST model to turbo-machinery simulations and found good agreement between the computations and the experimental data for all the considered cases. This model uses an “Automatic near-wall treatment” (Vieser et al. 2002, Esch et al. 2003, Menter et al. 2003a, b) that shifts gradually between the “Low-Reynolds-Number formulation” and the “Wall-Function Method”, based on the grid density. This wall modelling allows to reduce the computational effort as reported in studies of Menter et al. (2003a, b). Moreover, to reduce the excessive turbulence levels in regions with large normal strain (stagnation regions and regions with strong acceleration) the Kato-Launder production limiter (Kato & Launder, 1993) was selected in each simulation. All the CFD simulations were performed in a transient regime (Croquer et al. 2012) in order to take into account the fluctuations of different variables and the unsteadiness due to the rotation motion of the impeller. The frame change at the interface between the impeller and the steady domain was modelled using the unsteady sliding grid approach, also called Transient Rotor-Stator approach in

CFX (Ansys Inc. 2011). This model is robust and yields high accuracy predictions of the transient flow characteristics. Each of the RANS simulation was carried out for a total time of 1 sec with a time step of $2 \cdot 10^{-4}$ sec, which allowed to reach the convergence of the solution maintaining a Courant number lower than 1. The computational domain was discretized with ANSYS ® ICEM CFD™using a non-uniformly distributed unstructured mesh. The domain was divided into two parts: the rotating part, called rotor, and the steady part called stator. All meshes were generated according to a dimensionless wall distance y+ in the range between 50 and 100 as suggested in the analysis by Menter et al. (2003b). The total number of discretization elements was about 1,400,000: 1,000,000 in the rotor domain and 400,000 in the stator domain. The boundary conditions selected in the simulations are: a) Fixed value of the total pressure and a normal velocity to the boundary at the inlet section; b) Assigned constant value of the velocity at the outlet section (Ansys Inc. 2011). Preliminary simulation highlighted that the water flowing inside the whole turbine always maintains positive relative pressure and the air phase is missing. Thus, all the simulations were carried out considering a fluid made only by the water phase. 5 VALIDATION OF THE DESIGN PROCEDURE In order to validate the previous hydrodynamic procedure a Banki-Michell prototype was designed taking into account the following project input data: an energy drop ΔH of 10 m, with H_m = 35 m and H_v = 25 m; a water discharge Q = 0.03 m^3/s; a rotational velocity of the impeller ω equal to 750 rpm. The outer impeller diameter D_1 was first computed by solving Equations 1–3 along with the inlet velocity V and the specific energy drop E. Coefficient ξ was set equal to the mean value, ξ = 2.656 (Table 1), and C_V was set to 0.98. The optimal value of the outer diameter, D_1 = 134 mm, was estimated investigating seven values of the attack angle (α = 8° – 18°) through CFD analyses. The turbine width B was calculated equal to 77 mm according to Equation (4). The impeller geometry was then defined by investigating four numbers of blades N_b, from 30 to 60, and four diameter ratios D_2/D_1 ranging from 0.65 to 0.8.The CFD analysis was performed iteratively by testing a single parameter at time up to reach the optimum geometry. In each simulation the turbine performance was estimated as the ratio between the power supplied to the rotor, P_m, and the power lost by water passing through the turbine, P_h: where T is the torque, P_{inlet} and P_{outlet} are respectively the hydraulic power at the inlet and outlet section of the turbine. The analysis led to the follow optimal geometric parameters: α = 15°; N_b = 40; D_2/D_1 = 0.75. In Figures 3–5 the turbine efficiency is plotted versus the attack angle, number of blades and diameter ratio by keeping the other two parameters equal to their optimum value. Once the Banki-Michell prototype was designed the hydraulic efficiency was numerically tested taking into account different values of the flow rate, in the range between 0.010 m^3/s and 0.045 m^3/s.The numerical simulations were carried out considering a constant rotational velocity ω= 750 rpm. In Figure 6 a contour plot of the velocity field in the whole turbine domain and a detailed view of the impeller region are shown. The water velocity is uniformly distributed along the inlet surface of the impeller, maintaining an almost constant attack angle. At the outlet section of the impeller the velocity norm maintains low values, similar to the values at the outlet section of the turbine. As it can be seen in the plot the divergent section allows to properly reduce the velocity and to minimize the formation of turbulent structures downstream the impeller. The contour plot of the relative pressure in the whole domain and a detailed view of the impeller are reported in Figure 7. Figure 3. Optimization curve of the attack angle. Figure 4. Optimization curve of the number of blades. Figure 5. Optimization curve of the diameter ratio. The plot shows always positive relative pressure values, supporting the hypothesis of only one liquid water phase inside the impeller and the diffuser used in the CFD simulations. The efficiency of the turbine was computed for different hydraulic conditions corresponding to different discharge values. The efficiency values and the velocity ratio estimated in each simulation were reported in Figure 8 (the efficiency is shown beside each point). The plot shows that at the BEP corresponds to the design point of the turbine, Q = 0.030 m^3/s, with a hydraulic efficiency of 85%.The efficiency also maintains a value greater than 80% in the range between

Figure 6. Contour plot of the absolute velocity with a zoom

of the impeller region.

Figure 7. Contour plot of the relative pressure with a zoom

of the impeller region.

Figure 8. Efficiency curve of the designed turbine.

0.02 and 0.04 m^3/s, with the relative velocity V_r falling

in the range between 1.71 and 2.7. A strong reduction of the efficiency can be observed for low values of Figure 9. The estimated values of the specific energy at the outlet section of the designed turbine. Figure 10. The mechanical power P_m . the relative velocity, corresponding to low discharge values. The specific energy at the outlet section of the turbine, plotted versus the water discharge, is reported in Figure 9. The plot shows that at the BEP point the total head is the same considered in the design procedure, i.e. $H_V = 25$ m. In Figure 10 the mechanical power P_m is plotted versus the water discharge. On the basis of what has been shown so far, to improve the overall efficiency of the hydroelectric plant it is required to act on the velocity ratio V_r in such a way to have a value always close to that of the BEP point, $V_r = 2$. 6 ELECTRIC REGULATION OF THE IMPELLER ROTATIONAL VELOCITY In order to maintain a good velocity ratio in spite of the water discharge variability, it is possible to electrically regulate the rotational velocity of the impeller, to maintain a velocity ratio V_R always equal to 2. Using the same discharge values of the previous cases, the numerical simulations were carried out using the rotational velocity ω corresponding to the condition $V_R = 2$ in each hydraulic condition. Figure 11. The efficiency curves of the turbine in the two considered configurations, with and without regulation system. Figure 12. The estimated values of the specific energy at the outlet section of the designed turbine for the two considered configurations. In Figure 11 the efficiency curves of the turbine in the two configurations, with and without regulation system, are shown. Also, the rotational velocity required for each value of the tested flow rate is shown beside each point. RANS simulations were carried out in order to test the convenience and the feasibility of such solution. As it can be clearly observed the regulation system does improve the efficiency of the designed turbine, especially for low values of the water discharge. The regulation system allows to reach an efficiency peak of 86.8%, in any case the regulation system and greater than 84.3% for all the tested water discharge values. The values of the downstream specific energy H_V of the two configurations are compared in Figure 12. Observe that the control of the rotational velocity has not a significant effect on the total head at the turbine downstream section. Only for high values of the water discharge the downstream specific energy H_V is a bit lower in the configuration with the regulation system. To complete the analysis of the turbine with the regulation system, the mechanical power of the turbine in the two configurations has been compared. In Figure 13 the plot shows that the regulation system improves Figure 13. The mechanical power P_m of the designed turbine in the considered configurations. the turbine power production particularly for high values of the water flow. For example for the highest flow rate, $Q_{max} = 0.045\ m^3/s$, the mechanical power with the regulation system is $P_m = 7.7$ KW, while without regulation is $P_m = 6.0$ KW. In order to achieve an electric regulation system a back-to-back converter can be used. This system consists of two conventional pulse width modulated (PWM) voltage source inverters (VSIs). These inverters decouple the speed control of the electrical generator from the active and reactive power control implemented in the grid side converter. The latter converter is controlled in order to satisfy the grid regulations and maintain high quality electrical energy production, while the former ensures very flexible speed control of the electrical generator, tracking the maximum power point extracted from the hydraulic turbine (Consoli et al. 2010). This electrical regulation system can be coupled with a permanent magnet synchronous generator (PMSG) because of its high efficiency, high power density, low inertia and high power factor (Consoli et al. 2013). 7 CONCLUSIONS A new design procedure for a Banki-Michel turbine with positive pressure outlet flow has been applied to a test case and the performance of the resulting device has been numerically tested by means of CFD analysis. The results are quite encouraging, especially if an electric regulation system is used to control the impeller rotational velocity. In this case the efficiency of the turbine is always greater than 84.3% for a maximum/minimum discharge ratio equal to 4.5 and attains the peak 86.8% value for the minimum discharge value. The specific energy reduction between the inlet and the outlet section of course drops along with the discharge, almost independently from the electric regulation. This implies that it is not possible to maintain the same minimum specific energy value H_v downstream the turbine with different discharge values, by getting at the same time a good efficiency value of the

turbine. To this end, the use of hydraulic regulation is

mandatory.

Ansys Inc. 2011. ANSYS CFX Solver Modeling Guide – r. 14. Southpointe 275 Technology Drive Canonsburg, PA 15317.

Aziz, N.M., & Desai, V.R. 1993. A Laboratory Study to Improve the Efficiency of Cross-Flow Turbines (Engineering Report 1W-91). Department of Civil Engineering, Clemson University, Clemson, SC, USA.

Chattha, J. A., Khan, M.S., Wasif, S.T., Ghani, O.A., Zia, M.O., & Hamid, Z. 2010. Design of a cross flow turbine for a micro-hydro power application. Proceedings of ASME 2010 Power Conference, Chicago, Illinois, USA, July 13- 15, 2010, 637–644, Conference Sponsor: Power Division.

Consoli, A., Scarcella, G., Scelba, G. & Testa, A. 2010. Steady-State and Transient Operation of IPMSMs under Maximum Torque per Ampere Control. IEEE Transactions on IndustryApplications, vol. 46, n. 1, Jan–Feb 2010, pp. 121–129.

Consoli, A., Scelba, G., Scarcella, G. & Cacciato, M. 2013. An Effective Energy Saving Scalar Control for Industrial IPMSM Drives. IEEE Transactions on Industrial Electronics, 2013, vol. 60, n. 9, pp.: 3658–3669.

Croquer, S. D., De Andrade, J.,Clarembaux, J., Jeanty, F., & Asuaje, M. 2012. Numerical Investigation of a Banki Turbine in Transient State: Reaction Ratio Determination. Proceedings of of ASME Turbo Expo 2012: Turbine Technical Conference and Exposition, Volume 8: Turbomachinery, Parts A, B, and C. Copenhagen, Denmark, June 11–15, 2012. Conference Sponsor: International Gas Turbine Institute.

De Andrade, J., Curiel, C., Kenyery, F., Aguilln, O., Vásquez, A., & Asuaje, M. 2011. Numerical investigation of the internal flow in a Banki turbine. International Journal of Rotating Machinery, Hindawi Publishing Corporation,Vol. 2011, Art.ID.841214, 12 pages.

Esch, T., Menter, F.R. & Vieser W. 2003. Heat transfer predictions based on two-equation turbulence models. Proceedings of the 6thASME-JSMEThermal Engineering Joint Conference, March 16–20, 2003, Hawaii.

Integration of hydropower plant within an existing weir – “a hidden treasure”

M. Marence & J.S. Ingabire

UNESCO-IHE, Delft, The Netherlands

B. Taks

DEC Dommelstroom U.A. Sint-Michielsgestel, The Netherlands

ABSTRACT: Most of the best feasible hydropower sites in Europe have been developed and only half of

technically feasible potential is still available for development. Construction of each new hydropower plant in

Europe is loaded by very strong opposition and restrictive existing environmental and administrative procedures.

These restrictions caused nearly full stagnancy in erection of new hydropower plants with capacity above 10 MW.

Small scale hydropower potential is widely recognizable and development opportunities are significant, but

implementation struggles also on restrictions and negative image. High construction costs and low energy prices

in Europe, at the moment, additionally discourage new developers. Installation of new power plants in the

existing water structures without hindering their primary functions could become an attractive solution, saving

construction costs and minimized environmental impacts and negative public image. In this regards, the study of use of an existing weir structure developed for water management and navigation

purposes, has been developed. The weir structure on Dommel River, in Sint Michielsgestel, province of North

Brabant, the Netherlands has been constructed in 1970 is selected and analyzed in this study. A short overview

of the hydropower situation in the Netherlands including hydro potential, possible solutions and trends, tariffs

and legal regulations, and environmental and social constraints is also given.

1 INTRODUCTION

Hydropower, being large or small, remains by far

away the most important renewable energy being relied

on as it provides 19% of electricity worldwide and

17% of Europe. Most of the European hydropower

potential in terms of large scale are almost depleted or the few remaining can be hardly exploited due to environmental issues (Paish, 2002). Hydropower development in Europe is mostly only possible in the small hydropower sector (Marence in press) and also, these project developments are stalled and encum bered by rigorous environmental issues and also by low energy price levels in recent years in Europe. Development and increase of the hydropower pro duction in Europe are still possible. Rehabilitation, life extension, upgrading and optimization of exist ing hydropower facilities is an attractive way to take advantage of modern technologies and comprehensive planning, while minimizing environmental and social impacts is often linked with the extension of existing licenses. Development of alternative hydropower plants incorporated in existing hydro-technical structures represents an additional, attractive and lucrative solu tion. Such solutions use the existing hydro-technical infrastructure constructed and used for their primary function, but allow additionally generation of electrical energy. 2 ALTERNATIVE HYDROPOWER SOLUTIONS Incorporation of hydropower in the existing water infrastructure, where the electricity generation is not the primary priority is becoming an attractive solution for European hydropower. Several reasons make these solutions attractive for implementation. Social and environmental boundaries on existing hydro-technical structures are mostly known, defined and accepted. Low energy prices and feed in tariffs does not simulate the hydropower market and cheaper solutions are needed. Most of the hydropower infrastructures are linked with high construction and implementation costs and use of existing infrastructure gives the possibility for economically attractive solutions. Additionally, new definition and modification of the existing structures for additional purposes gives the possibility for environmental and social improvements, making the solutions additionally attractive. Identification and use of such non-traditional solutions can be seen as searching for the hidden treasure. The solutions could be found all around us, in municipal and agricultural water systems, existing dams and Figure 1. Water infrastructure suitable for additional hydrpower use. hydropower plants and also other hydraulic circulation systems. Some of the possible solutions are shown in Figure 1. 2.1 Municipal and agricultural water systems The municipal systems have a strong potential in drinking water supply but also in sewage systems. The potential is stronger in mountainous or hilly regions, where water intakes are situated at higher altitude and access pressure must be reduced before consumption. This reduction is performed by energy dissipation in specially constructed valves saving system pipes from bursting and permitting the operational use of water. The pressure reduction by drinking water turbines is often used in Alpine area. The turbines installed in the transmission pipelines are technically easier compared with a distribution network because flow and pressure in transmission pipelines are less fluctuating. The advantage of the drinking water system, used for energy production, is in existing closed system with all surrounding facilities that could be fully used. The drinking water turbines must satisfy strong sanitary requirements without any influence on the water quality and must be built on redundant way allowing full water supply at all times independent of the turbine operation. Bölli & Feibel (2015) described their experience with such systems and implementation possibilities in the developing world. In the case of sewage water, the possible natural head between the residential area and the outlet of treated water in nature could be used. Use of untreated or treated water is possible and depends on the topologic setup. In the case of untreated water, the energy system must be able to deal with all kinds of generated rubbish. Irrigation systems are mostly built with a function to maximize the irrigated area and transport water as far as possible. Therefore, any energy extraction from the system reduces the primary function. In some special cases, cascades in the system, these denivelations could be used for energy generation. 2.2 Dams, hydropower and other plants Dams are storing water for different uses and requirements. More than 70% of world dams are built for a single purpose and irrigation is the most common use. Just 17.4% of world dams are built for hydropower (ICOLD, 2007) and energy generation. The dams built for other purposes give the possibility for additional energy generation. The potential and charm of such solutions could be seen in the case of USA where more than 80.000 non-powered dams have been detected with a total potential of additional 12 GW (US DOE, 2012). Also existing hydropower plants give the possibility for additional upgrade that could be done during refurbishment by installation of modern equipment with higher efficiency, higher load factor, but also implementing accompanying ecological measures, such as fish ladders adopted on local fish population or ecological minimal flow turbines. Another possibility is given by the installation of the hydropower plants in the existing or new ship navigation locks. Process water used for filling and emptying of ship locks could be used for energy generation, but also, the weir controlling navigation levels could be used for energy generation as in the case of the navigation system on river Waal in the Netherlands where three navigation weir structures including hydropower plants have been built. Also, the fish bypass systems could be included in the energy generation, especially usage of the water used to help fish to locate and navigate their way to the fish pass entrance. 2.3 Hydraulic circulation systems Similar as in the drinking water systems, industrial cooling or heating systems can result in a pressure excess that can be recovered over the hydro turbines instead of energy dissipaters. Some of the desalination plants use reverse osmosis to separate water from dissolved salts through semipermeable membranes under high pressure (from 40 to 80 bars). The residue of water containing salt, still at high pressure could be passed through a turbine in order to recover part of the energy used for the initial compression. 3 IMPLEMENTATION IN EXISTING WEIR STRUCTURE 3.1 Weir sint michielsgestel This study focused on the existing weir located in Sint Michielsgestel, North Brabant at Dommel River in the Netherlands. The weir was initially built in 1970 to keep the upstream water level constant at 4.6 m with a normal head difference of 1.8 m. The downstream water level and therefore the denivelation head are a function of discharge. The discharge over the weir is automatically regulated by flap gates situated in three weir openings of 5.0 m each. The weir has been built to regulate the discharge and the water levels in the surrounding channels and is operating fully automatic.

Figure 2. Upstream view on the existing weir structure.

Figure 3. Downstream view on the existing weir structure.

Adaptation of the system has been performed in 2006 when the fish passage channel situated approximately 100 m upstream the weir structure has been built. The fish passage channel with 1.6 m^3/s discharge is built as a natural river bed with stone steps forming pools and heterogeneous environment favorable for fish migra tions. Construction of the fish passage was necessary for fulfilling the European Water Framework directive requirements. Figure 2 shows the upstream and Figure 3 the downstream view of the existing weir structure. The constant denivelation and discharge give an ideal condition for installation of a small hydropower plant. The private investor started the licensing pro cess. During the licensing process, UNESCO-IHE has prepared a technical study checking and optimizing the site (Ingabire, 2014).

3.2 Hydrology and power estimation

The water level and the discharge over the weir are measured automatically and the hydrological data were collected from the local water governance authority. Daily data for 34 years (1977–2011) have been used for the study. The automatic hourly upstream and downstream water level measurements together with discharge measurements result in a very good correlation between the discharge and available head which was Figure 4. Flow duration curve (FDC) for selected years with minimal, maximal and mean values. used for the energy simulations. The total head is kept nearly constant at 1.80 m up to the discharge of 15 m 3 /s and drops semi-parabolic to less than 1.00 m for discharges beyond 32 m 3 /s. Additional hydraulic simulation have been performed to find the capacity of the existing weir structure under flood conditions. The flood load on the site has been changed with performed flood protection measures in the upstream area and maximal 100 years flood of 52 m 3 /s have been defined. The discharge capacity of each gated bay is over the 30 m 3 /s. These improved flood control measures lead to overestimated discharge of the weir and possibility to reduce the weir width – two weir fields needed in case of flood, giving possibility to construct the new power plant in one of the weir fields. 3.3 Selected solution The site at Sint Michielsgestel is very favorable for small hydropower plant based on several criteria. The weir is situated close to existing infrastructure which enables simple and economical grid connection. Existing weir structure forming a head difference and pre-defined regime of operation simplifies the licensing process and also minimize the influences of the plant structures in the neighborhood. Existing fish passage reduces the environmental impact (no change) and also the construction costs and effort. Existing infrastructure (access road) on the weir additionally simplify the construction and operations. The study (Ingabire, 2014) included all of the influenced parameters and tries to find an optimal solution and the optimal setup for the planned power plant. Multi-criteria analysis has been performed for selecting optimal power plant setup and also optimal turbine type. Following criteria have been included in the analysis: – Run-of-river system without any storage possibility (fixed upstream water level) – Design discharge of 10 m 3 /s (75 days mean discharge) is selected – Head of 1.0–1.8 m determines application of very low head turbines Figure 5. Power production based on characteristic flow duration curve (FDC) for 2007 with minimal, maximal envelope and mean values. – Wide turbine operational discharge range – Permanent land occupation on both weir banks is limited. Considering all of these criteria, the solution with a power plant merged in one of the weir fields is suggested and selected. Different types of low head turbines have been analyzed with special consideration on construction opportunities, efficiency, flexibility and construction costs. Based on given boundaries and restrictions, the solution with a screw turbine has been seen as the most promising option. The turbine can be installed in one of the weir fields with minimal reconstruction works. The lower efficiency and therefore power output compared to some other turbine types could be compensated with lower turbine construction costs and simpler construction and large turbine flexibility in head and especially discharge variation. Therefore, the screw turbine was suggested as an optimal solution. Based on the selected screw turbine, simulations of the energy production have been performed and the production is estimated for the simulated period 1977–2008. All the simulations included the ecological flow over the fish package of 1.6 m 3 /s. Simulations have been made on the daily and hourly basis. Power output based on characteristic flow duration curves and for the year 2007 are presented in Figure 5. Based on simulation annual energy product “on wire” varies from 482 MWh to 900 MWh. The mean energy production of 740 MWh characterizes plant capacity factor of the 58.2% and could be seen as an optimal plant size. The mean annual energy production of the plant is enough to supply energy for approximately 160 European households. 3.4 CONSTRUCTION The construction of the project with the screw turbine will start inMay 2016 and is planned to be finished in September 2016.A screw turbine with a 10 m 3 /s design discharge will be implemented.The screw with a diameter of 4.0 m will be installed in the middle field of the existing weir. The symmetrical setup was required by Figure 6. Photomontage of the finished project in operation. the authorities to achieve symmetrical flow and soil erosion. To ensure water drainage at high flow rates, the inlet of the hydro screw is not directly situated at the existing weir. Between the weir and the turbine a 6-meter long overflow channel with a height which is just above the target level is planned. In the case of emergency when the upper water level rises, the excess water can flow over the side of the walls of the channel. The hydro turbine will be founded on a sheet pile wall and connected with the existing weir structure. The hydro screw turbine and generator are fabricated as a self-supporting system and will be pre-made at the factory. The foundation for the upper bearings and output unit is planned on reinforced underwater concrete. The lower bearing of the screw will be founded on a strip foundation in the river. The plant will be connected directly, without any transformers on the low voltage (230 V) local grid with and 3*250 ampere connection. The variation in head and especially in discharge will be regulated by the variable speed generator. This project is the first project in the Netherlands where a local water department has given a private organization the rights to build a hydro power installation in the middle of a river, attached to an existing structure. The everlasting concession is given to the site. The concession can be quitted only in case of special hydrological needs and is only limited with the lifetime of structure, that is assumed by 25+ years. No special measures, except a fence around the weir and the inlet screen are foreseen against vandalism on the fully automatic site. Some specific requirements from the local water department – the use of the middle gate of the weir, the 6-meter long overflow channel and the reinforcement of the surface around the output of the screw – provided extra costs. Also, the crowdfunding (legal fees and marketing) made the project more expensive. The total project costs is estimated at one million euros. These costs include the turn-key building of the hydro screw, all civil engineering works, the grid

Figure 7. Longitudinal section through the powerhouse.

connection, feasibility studies and engineering, legal

fees, project management and fund raising costs.

3.5 Financing model

The project is for 75% financed by crowdfunding; the

rest of the investment includes a contribution from

local associations. Almost 500 participants from the

region bought certificates in the cooperation. Partic

ipants get their return in free power and dividends.

The expected return for the crowd funders is estimated

between the 4 and 8% (IRR). The free power gives the

(mostly) private investors an additional VAT benefit

which is a result of an investment in equity that is

VAT free. Because no bank debt is used is this case,

there will be no short-term cash flow issues to repay

the debt.

4 CONCLUSIONS

Hydropower rehabilitation and optimization together

with the installation of the power plants in existing hydraulic structures, a “hidden treasure”, needs to be tapped. There is a considerable potential for rehabilita tion, life extension, upgrading and optimization of existing hydropower facilities. Tapping this potential would contribute improving the energy efficiency of hydropower, guarantee the safety of aging plants and produce substantial economic gains. Rehabilitation and upgrading of existing plants can be made taking advantage of modern technologies and comprehensive planning while minimizing environmental and social

Special session: Buoyancy-driven flows

Chen, S., Lei, C. & Patterson, J.C. 2014. Numerical modeling of a concurrent PIT/PIV experiment with TLC particles in a reservoir model subject to periodic thermal forcing. Numerical Heat Transfer, Part A: Applications 66(1): 64–88.

Coates, M.J. & Patterson, J.C. 1993. Unsteady natural convection in a cavity with non-uniform absorption of radiation. Journal of Fluid Mechanics 256: 133–161.

Coates, M.J. & Patterson, J.C. 1994. Numerical simulations of the natural convection in a cavity with nonuniform internal sources. International Journal of Heat Mass Transfer 15(3): 218–225.

Farrow, D.E. 2013. Periodically driven circulation near the shore of a lake. Environmental Fluid Mechanics 13: 243–255.

Lei, C. & Patterson, J.C. 2006. Natural convection induced by diurnal heating and cooling in a reservoir with slowly varying topography. JSME International Journal (B) 49(3): 605–615.

Lowe, R.J., Shavit, U., Falter, J.L., Koseff, J.R. & Monismith, G. 2008. Modeling flow in coral communities with and without waves: A synthesis of porous media and canopy flow approaches. Limnology and Oceanography 53(6): 2668–2680.

Monismith, S., Imberger, J., & Morison, M.L. 1990. Convective motions in the sidearm of a small reservoir. Limnology and Oceanography 35: 1676–1702. Nepf, H.M. & Oldham, C.E. 1997. Exchange dynamics of a shallow contaminated wetland. Aquatic Sciences 59: 193–213. Patterson, J.C. 1984. Unsteady natural convection in a cavity with internal heating and cooling. Journal of Fluid Mechanics 140: 135–151. Shin, J.O., Dalziel, S.B. & Linden, P.F. 2004. Gravity currents produced by lock-exchange. Journal of fluid mechanics 521: 1–34. Tsakiri, M. & Prinos, P. 2016. Microscopic simulation of unsteady natural convection in a partially vegetated reservoir induced by absorption of radiation. (to be submitted to Journal of Hydraulic Engineering). Tsakiri, M., Prinos, P. & Koftis, H. 2016. Numerical simulation of turbulent exchange flow in aquatic canopies. Journal of Hydraulic research (in press) doi: http://dx.doi.org/10.1080/00221686.2016.1141803. van Gent, M.R.A 1995. Wave interaction with permeable coastal structures. Delft: PhD Thesis, Delft University. Whitaker,

S. 1996. The Forchheimer equation: a theoretical development. Transport in Porous Media 25: 27–61. Zhang, X. & Nepf, H.M. (2009). Thermally driven exchange flow between open water and aquatic canopy. Journal of Fluid Mechanics 632: 227–243. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Experiments on the impact of snow avalanches into water

G. Zitti

Departement of Civil and Building Engineering and Architecture

Università Politecnica delle Marche, Ancona, Italy

C. Ancey

Faculté de l'Environement Naturel, Architectural et Construit

Ecole Polytechnique Fèdèrale de Lausanne, Lausanne, Switzerland

M. Postacchini & M. Brocchini

Departement of Civil and Building Engineering and Architecture

Università Politecnica delle Marche, Ancona, Italy

ABSTRACT: The number of anthropized reservoirs threatened by snow avalanches is steadily increasing and

impulse waves caused by avalanche impact are becoming a considerable risk for such basins. The dynamics of

the impact of a snow avalanche into a water body is studied through laboratory experiments, where a granular

material, with solid density slightly lower than that of the water, is used to simulate the buoyant behaviour of

the snow. The proposed experimental model shares many similarities with those used to investigate the impact

of landslides into water bodies (e.g., see Fritz et al. 2003a, Fritz et al. 2003b), but it also clarifies the differences

between the impact of an avalanche and that of a landslide: while a landslide typically reaches the bottom of the water body, because of its high constant density, in the present experiments a floating motion of the impacted mass has been observed. The wave generation and its propagation are acquired by high-frequency cameras placed along the flume. Some preliminary results on wave amplitude/height decay and on wave celerity state the strongly non-linear behaviour of the generated wave. Further, their comparison with the corresponding predictive relations proposed for the case of landslides (Heller and Hager 2010) highlights a different wave decay and celerity evolution in the proximity of the impact.

1 INTRODUCTION

The increasing anthropic pressure on high-altitude areas and sub-polar regions has led to new hazard situations. One of these is the exposition of human settlements to snow avalanches, which has promoted research on snow rheology and avalanche dynamics. One of the most recent problems is the generation of tsunami-type waves due to the avalanche impact into water basins, and the subsequent propagation and run up. Over the last decade, a number of events occurred of snow avalanches hitting water basins, such as lakes in the alpine area and fjords in the sub-polar areas. The analysis of impact waves generated by snow avalanches entails a number of dynamics: the avalanche formation and detachment, its propagation down the slope, its impact into water, the subsequent wave formation and, finally, the wave propagation and eventual run-up. These aspects can be studied separately: the avalanche formation and propagation are closely related to meteorological conditions and terrain characteristics and, at the moment, several statistical fluid-dynamics models are available for predicting avalanche run-out distances, flow velocities and impact pressures (e.g. Lied and Toppe 1988, Gruber and Bartelt 2007, Christen et al. 2010, Ancey 2012), while additional insights have been brought by small-scale laboratory experiments (Savage 1989, Hutter 1996); the wave propagation and run-up is a well known topic and recent numerical applications in enclosed basins (Couston et al. 2015) have shown that the wave cannot disperse and is reflected back, therefore the interaction with incoming waves could significantly increase both the run-up and its destructive effects; the impact and the wave formation (near field) are strongly related, hence they cannot be studied separately. The literature on the topic of the impact and the wave formation is recent (Evette et al. 2011, Naaim 2013). On the other hand, the problem shows similarities with the topic of impulse waves generated by the impact of landslides into water, which has been significantly studied (Noda 1970, Kamphuis and Bowering 1972, Fritz et al. 2003a, Fritz et al. 2003b, Fritz et al. 2004, Heller 2007, Mohammed and Fritz 2012). However, the important differences suggest that specific analyses of the impact of snow avalanches in water basins be made. The main differences between snow avalanche and landslide dynamics are related to the density (both solid and bulk densities) of the material. First, the volume (i.e. the bulk density) of the snow avalanche can vary during the slide, while the volume expansion of a landslide is approximately 20% (Heller 2007). Moreover, because of the solid density of the snow, the mass rebounds in the water and moves close to the water free surface, while the motion of a land slide follows the slope even below the free surface, until it comes to rest, because of either the friction down the slope or the impact with the bottom. Such dif ferences suggest that the characteristics of the impulse waves generated by snow avalanches are different from those of the impulse waves generated by landslides, especially in the near field. A small-scale model, proposed in Zitti et al. (2016),

is here used to simulate the impact of snow avalanches into water. Such model represents a two dimensional approximation of the problem and is similar, but not equal, to the model used for landslides, because: 1) it allows some volume expansion of the mass down the slide and 2) the mass is characterized by a solid density lower that the water density, this making the mass buoyant after the impact. The experimental setup and the acquisition of the characteristics (amplitude, height and celerity) of the impulse waves simulated in experiments are described in Section 2. After that, a brief description of the results on the water waves generated by landslides from literature is reported in Section 3. The acquired data, i.e. the amplitude decay, the height decay and the celerity evolution in space, are reported in Section 4, where they are compared to the predictive functions of the characteristics of the impulse waves generated by landslides found in literature. Some conclusions close the paper.

2 PHYSICAL MODEL AND EXPERIMENTS

The physical model used to simulate an avalanche impact in water (Figure 1) represents a two dimensional approximation of the problem. We con sider a slope with inclination α, entering a two dimensional water body, whose initial still water depth is denoted by h. A global coordinate system is used (x, y) with origin placed at the still water shoreline, the x-axis being positive streamwise and the y-axis downward. A mass M , with initial bulk density $\rho_{m,rest}$, is released at a distance '' s from the shoreline and slides down the slope, driven by gravity (of acceler ation $g = 9.81$ m/s ^{2}). During the slide along the chute, the average bulk density $\rho_m(t)$ and the shape s(t) of the avalanche vary with time as a result of volume expansion. The impact on the water surface generates an impulse wave, whose surface is described by the function $\eta(x, t)$. The experimental setup that reproduces such mod els is composed of a wooden chute, with slope angle $\alpha = 30^\circ$, reaching the bottom of a 3 m long pris matic flume filled with water, with transparent glass sidewalls. Both chute and flume are 0.11 m wide. Figure 1. Two-dimensional physical model for an avalanche impacting into water. The water is coloured with methylene blue and the used still water depth is 0.14 m. The avalanche mass M is composed of granular expanded clay. This material is chosen because its bulk density at rest was 489 kg/m 3 , with a standard deviation of 8%, which is a good approximation of the compact snow density before the avalanche detachment. Furthermore, the use of a granular material allows the volume to expand down the slope. Finally, the solid density ρ s of the material is slightly smaller than the water density ρ w , this allowing the material to float in the water body. The average grain diameter of the used material is d g = 9 mm.The mass is placed at the top of the slope, with distance '' s from the shoreline, and is released, thus sliding down the slope and reaching the flume, where it enters the water body and generates the wave. To acquire the dynamics of the impacting mass and of the wave, three cameras have been placed along the flume, with the optical axis perpendicular to the flume length. One is an high-speed camera, located in front of the shoreline/impact zone, acquiring 256 × 785 px images, with a frequency of 1000 fps. Further, two low-speed cameras have been placed along the flume, collecting images of the free water surface with a resolution of 120 × 658 px and a frequency of 120 fps. A sample of the impact is shown in the sequence of Figure 2. Following the image editing procedure described in Zitti et al. (2015), the snapshot of the high-speed camera is used to estimate the average velocity of the slide at impact. The aerial part of the avalanche is considered and the image sequence is divided into 30 time intervals. For each interval, the displacement of the moving particles at impact is measured. For each analysed sample the mean value of the particle displacement is evaluated and the impact velocity is then estimated. The obtained impact velocity shows a consistent mean value and a relatively small standard deviation, hence the impact velocity can be assumed to be constant over time and equal to the mean value of the particle velocities v m , whose direction is parallel to the sloping ramp. Further, both the high-speed camera (at the impact zone) and the low-speed cameras (along the flume) have been used to extrapolate the water elevation η(x, t) of the generated impulse wave. Each frame of

Figure 2. Snapshot of the impact for experiment B, charac

terized by M = 300 g, '' s = 0.66 m and v m = 1.507 m/s.

the videos has been first cropped to avoid environmen tal disturbances, the contrast has been increased and the grey-scale image has been converted into a black and-white image. From the resulting image, where the water is black, the level of the free surface was extracted. The difference between the obtained sur face level and the still surface of the first frame gives the instantaneous water elevation $\eta(x, t)$. Four experiments are analysed in the present paper.

The variation of the mass and of the slope length affects the velocity at the impact. Such avalanche charac teristics of each experiment are reported in Table 1. The generated waves are characterized by a leading wave, where the maximum amplitude occurs, and a complicated multi-modal wave train. The time when the wave reaches the end of the flume, i.e. when the first elevation occurs there, is denoted with t_{lim}. For $t \geq t_{lim}$ the acquired wave is sensitive to reflection. The decay of wave amplitude $A(x)$ and wave height $H(x)$, as well as the local values of the wave celer ity $c(x)$, have been computed from the instantaneous water elevation $\eta(x, t)$, obtained from the processing of images acquired by both the high-speed camera and the low-speed cameras. Not all the flume length has been analyzed: part of the zone where the impact occurs has been overlooked to avoid the impacting and floating particles. Further, if the end of the leading wave t_{end},

i.e. the first zero up-crossing after the maximum ele

vation, occurs in a portion of the wave that is prone to

wave reflection due to the flume end (i.e. $t_{end} > t_{lim}$),

the signal is not analysed. Table 1. Experimental avalanche characteristics.

ID	M [g]	" s [m]	v_m [m/s]
A	200	1.21	2.052
B	300	0.66	1.507
C	500	0.66	1.434
D	600	0.66	1.438

The amplitude decay and the height decay have been evaluated considering the wave acquired at each distance x from the shoreline. Thus, the resolution of the distance corresponds to the resolution of the used cameras. The amplitude has been evaluated as the maximum value attained by the water elevation, while the height has been evaluated using the minimum elevation within the first trough after that crest. To evaluate the celerity evolution the space domain under investigation has been divided into intervals with dimension of δx = 25 mm and mid location x_i. The water elevations at the edges of the i-th interval, labelled as $\eta(x_i - \delta x/2, t)$ and $\eta(x_i + \delta x/2, t)$, have been used to evaluate the celerity of the wave inside such interval as follows. The time δt_i needed by the wave to travel from $x_i - \delta x/2$ to $x_i + \delta x/2$ has been evaluated, using a cross-correlation procedure (Box et al. 2011), as the time delay that maximizes the cross covariance function between the two waves at the edges of the interval. The cross-correlation procedure has been applied considering only a portion of the wave: from the beginning of the acquired data up to the time of the first zero-up-crossing following the wave crest, labeled as t_{end}. If t_{end} was greater than the time when the first water elevation appeared at the end of the flume t_{lim}, the cross-correlation procedure was applied to the portion of the wave before t_{lim} (instead of t_{end}), thus avoiding the interferences of reflection. When the first zero-down-crossing time was larger than t_{lim}, the remaining signal was too small to made a reliable cross-correlation procedure and the analysis was interrupted. Those cross-correlation procedures that were characterized by poor covariance functions have been discarded. Then, the celerity of the wave at point x_i was evaluated as the ratio between the space traveled by the wave and the time δt_i: 3 PREDICTIVE RELATION FOR LANDSLIDE GENERATED IMPULSE WAVES The literature on water waves generated by landslides is rich, because of the large number of destructive events. For example, the Lituya Bay

(Alaska) experienced at least four giant impact wave events over the last two centuries (Miller 1960), while in the Vajont artificial reservoir (Italy) a landslide-induced impact wave overtopped the dam, destroyed the city of Longarone and killed 2000 people. The first studies (Noda 1970, Kamphuis and Bowering 1972) used a simplified, bidimensional, model of the problem and suggested that the volume and velocity of the land slide at impact are the key parameters for the generated leading wave. Further works (Huber 1980, Huber and Hager 1997) showed the importance of other parame ters, like the material density and the slope angle. Since a similar model has been used in the present work to simulate snow avalanches, the main literature results on two-dimensional models with granular slides are reported. The more advanced two-dimensional model with granular material was developed first by Fritz (2002), which used a pneumatic landslide generator that allowed to vary independently many landslide characteristics which affect the impulse wave genera tion. The physical model was composed of a box filled with granular material, that was accelerated with the mentioned pneumatic landslide generator (Fritz and Moser 2003) along a 3 m long sloping ramp. When the box reached the maximum velocity, it released the mass and decelerated, while the granular slide mate rial left the box and accelerated further down the ramp until it reached the water body, generating an impulse wave. The water body was composed of a rectangular prismatic water wave channel, 11 m long, 0.5 m wide, and 1 m deep. The wave and slide characteristics were recorded by means of different measurement techniques. In particular, the water free surface was monitored by means of seven gauges along the channel main axis (1 m spacing). In Fritz (2002) 137 experiments were conducted varying the still water depth, the avalanche thickness, the slide impact velocity, the bulk slide volume and the initial (in the box) slide shape. The same physical model was used for the 86 experiments reported in Zweifel et al. (2006), where also the slide bulk density was varied and two different grain diameters were used, and for the 211 experiments reported in Heller (2007), where, among the other slide characteristics, also the slope angle was varied. In Heller (2007) the data obtained by Fritz (2002) and Zweifel et al. (2006) was used to achieve several empirical equations for the main wave characteristics, most of them depending on the impulse product parameter P, defined as:

where $Fr = v_m/(gh)^{1/2}$ is the slide Froude number, $S = s/h$ is the relative slide thickness, $M^* = M/(\rho_w b h^2)$ is the relative slide mass and α is the slope angle. The scaling parameters were the still water depth h, the

flume width b, the gravitational acceleration g and the

water density ρ_w .A theoretical analysis on the parame

ter P was proposed by Heller and Hager (2010), where

it was related to the slide momentum flux, following

the theory of Zweifel et al. (2006). In particular, the maximum wave amplitude and height were predicted by means of: with $R^2 = 0.88$ and deviation ±30%, and with $R^2 = 0.82$ and deviation ±30%. Such equations are valid for the dynamics occurring in the impact zone, i.e. for $x^* < x^*_m$, where $x^* = x/h$ and $x^*_m = (11/2)P^{\frac{1}{2}}$. Further, for $x^* > x^*_m$ the wave amplitude decay was predicted by means of: with $R^2 = 0.81$ and deviation ±30%. Finally, the wave height decay was predicted by means of: with $R^2 = 0.81$ and deviation ±30%. The wave celerity c has been evaluated in Heller (2007) as the average celerity between the first gauge and the last gauge not affected by wave reflection in the flume. The celerity of the primary crest locations c_{am} has been related to the maximum amplitude of the wave, obtaining the following empirical relation: Such equation represents an approximation for the solitary wave celerity, reported in Heller and Hager (2010). In fact, as already suggested in Kamphuis and Bowering (1972) and Huber and Hager (1997), the celerity of a landslide-generated impulse wave can be well approximated by the solitary wave celerity $c = \sqrt{g(h + a)}$. However, the celerity proposed for landslide-generated impulse waves is not the local value, but its average value along an almost entire flume. In the following section the amplitude decay and the height decay observed in our experiments are compared with the predictive relations obtained for landslide-generated impulse waves. Further, the celerity obtained from the data analysis is compared with the celerity of the solitary wave. 4 RESULTS In this section the wave decay and the celerity evolution are analysed.To compare them with the predictive relations from literature on landslide-generated impulse

waves they are reduced to dimensionless form.The val

ues of the wave amplitude decay A(x) and wave height

decay H (x) are made dimensionless dividing by their maximum value, while the distance is made dimen sionless through the still water depth. Then, the results of the experimental analysis $A(x*)/A_m$ and $H(x*)/H_m$ are reported in Figure 3. The results obtained from the high-speed camera (red dots) and the low-speed cameras (blue dots) are reported. The wave ampli tude decays obtained from the two acquisition systems show a rather good correspondence. The gap that sometimes appears between the two data is due to the different resolution of the two types of cameras, while the jumps within each datum are due to the resolution of the acquisition system (small jumps for the low speed cameras along the flume) or to the changing of leading wave. Concerning the wave height, such errors are emphasized by the large variability of the trough. In particular, the strongly multi-modal shape of the wave induces a continuous variation of the trough, that sometimes disappears, allowing the presence of frequent and large jumps in the datum. With refer ence to literature relations, dividing equation (5) by (3) and equation (6) by (4), gives the same relative decay expression for both the wave amplitude and the wave height: The predictive relation (8) is also reported in Figure 3 to compare it with the experimental data (black line). The decay of the amplitude and of the height of the experimental waves occurs already for $x* \le x*_m$. Thus, the experimental decay is larger than that predicted by literature relations for landslide-generated waves. The wave celerity, evaluated as described in section 2, is made dimensionless by dividing through $\sqrt{gh}$ and reported in Figure 4, where results from both the high speed camera and low-speed cameras are reported.The reliable data are reported with black markers. Data are taken as unreliable if the maximum cross covariance function (XCF_m) is smaller than 0.9. Further, if the difference between XCF_m and the value of XCF corre sponding to the overlap of the two waves without time delay is smaller than 0.01, the two compared waves are too close in time to obtain a good cross-correlation. Furthermore, wave reflection at the end of the flume prevents the determination of the celerity over a large portion of the flume. Although part of the data is not reliable an increasing behaviour up to the celerity of shallow-water waves is evident in all four cases. Using the same scaling, the celerity of a solitary wave, proposed in literature for the case of landslides, is:

The experimental celerity acquired along the flume

A(x) is used to evaluate the celerity of the solitary wave

Figure 3. Wave amplitude decay (top panel) and wave height decay (bottom panels) acquired from the high-speed camera (red dots) and from the low-speed cameras (blue dots), compared with equation (8) (black line).

Figure 4. Celerity: the circles represent the celerities

obtained from the high-speed camera in the near field zone,

while the squares represent the celerities obtained from the

low-speed cameras along the flume. The black markers are

Fritz, H. M., W. H. Hager, & H.-E. Minor (2004). Near field characteristics of landslide generated impulse waves. Journal of Waterway, Port, Coastal, and Ocean Engineering 130(6), 287–302.

Fritz, H. M. & P. Moser (2003). Pneumatic landslide generator. International Journal of Fluid Power 4(1), 49–57.

Gruber, U. & P. Bartelt (2007). Snow avalanche hazard modelling of large areas using shallow water numerical methods and gis. Environmental Modelling & Software 22(10), 1472–1481.

Heller, V. (2007). Landslide generated impulse waves: prediction of near field characteristics. Ph.D. thesis, ETH Zürich, Zurich.

Heller, V. & W. H. Hager (2010). Impulse product parameter in landslide generated impulse waves. Journal of waterway, port, coastal, and ocean engineering 136, 145–155.

Huber, A. (1980). Schwallwellen in Seen als Folge von Felsstürzen. Ph. D. thesis, ETH Zürich. in German.

Huber, A. & W. Hager (1997). Forecasting impulse waves in reservoirs. In XIX Transactions of the International Congress on Large Dams, Rio de Janeiro, Brasil, Volume 1, pp. 993–1006.

Hutter, K. (1996). Avalanche dynamics. In V. P. Singh. (Ed.), Hydrology of disasters (Kluwer Academic Publications ed.)., pp. 317–394. Springer.

Kamphuis, J. & R. Bowering (1972). Impulse waves generated by landslides. In Proc. 12th Coastal Engineering Conf., ASCE, New York, pp. 575–588. Lied, K. & R. Toppe (1988). Calculation of maximum snow-avalanche run-out distance by use of digital terrain models. Annals of Glaciology 13, 164–169. Miller, D. J. (1960). Giant waves in lituya bay, alaska. Technical report. Mohammed, F. & H. M. Fritz

(2012). Physical modeling of tsunamis generated by three-dimensional deformable granular landslides. Journal of Geophysical Research: Oceans (1978–2012) 117(C11). Naaim, M. (2013). Impulse water waves generated by snow avalanches. In International Snow Science Workshop (ISSW), pp. 619–624. Irstea, ANENA, Meteo France. Noda, E. (1970). Water waves generated by landslides. Journal of the Waterways, Harbors and Coastal Engineering Division 96(4), 835–855. Savage, S. B. (1989). Flow of granular manalyses.Theoretical and Applied Mechanics, 214–266. Zitti, G., C. Ancey, M. Postacchini, & M. Brocchini (2015). Impulse waves generated by snow avalanches falling into lakes. In E-proceedings of the 36th IAHR World Congress, The Hague, the Netherlands. Zitti, G., C. Ancey, M. Postacchini, & M. Brocchini (2016). Impulse waves generated by snow avalanches: Momentum and energy transfer to the water body. Journal of Geophysical Research. (submitted). Zweifel, A., W. H. Hager, & H.-E. Minor (2006). Plane impulse waves in reservoirs. Journal of waterway, port, coastal, and ocean engineering 132(5), 358–368. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Two dimensional Lattice Boltzmann numerical simulation of a buoyant jet

A. Montessori, P. Prestininzi & M. La Rocca

Dipartimento di Ingegneria, Università degli Studi Roma Tre, Roma, Italy

D. Malcangio & M. Mossa

Dipartimento di Ingegneria Civile, Ambientale, del Territorio, Edile e di Chimica, Politecnico di Bari, Bari, Italy

ABSTRACT: Since its introduction (Succi, 2001), the Lattice Boltzmann Method (LBM) has obtained increas

ing attention in the field of Computational Fluid Dynamics, due to its intrinsic simplicity and ability even in

dealing with rather complex flows. This work is aimed at assessing the ability of the two dimensional version

of the LBM in simulating a buoyant jet. The latter is a saline jet, entering a uniform flow. The inlet velocity of

the jet is perpendicular to the velocity of the flow. The adopted two-dimensional Lattice Boltzmann formulation

is equivalent to the Navier-Stokes equation with a Boussinesque gravity force term. Experiments on turbulent

negatively buoyant jets into cross flow have been utilized to assess the validity of the numerical simulations.

A conductivity probe for high resolution measurement of turbulent density was utilized to measure the mixing

of the salt water into the cross current, whereas the Acoustic Doppler Velocimeter (ADV) system was used for

the measurement of the instantaneous threeflow velocity components. The comparison of the two dimensional

numerical results with the experimental results shows that the former are able to capture the general appearance

of the flow, although there are important differences. In particular, two dimensional simulations are characterised

by vortices which do not appear in experiments: this is probably due to two-dimensional nature of the numerical

simulation and its reduced ability in dissipating turbulent structures.

1 INTRODUCTION

Since its introduction in the Computational Fluid

Dynamics, the Lattice Boltzmann Method (LBM) has

gained increasing popularity. An exhaustive review

of the state of the art, as well as of the variety of

the applications can be found in Succi (2001) and

Aidun & Clausen (2010). The reason of such a rapid

diffusion is due to the intrinsic simplicity of the numer

ical algorithm, which is linear and local, and to its

extreme versatility, being able to simulate very differ ent kind of flows while keeping the same structure.

Very briefly, the LBM describes the evolution of the flow in terms of a discrete set of Probability Den sity Functions (PDFs), which give the probability at a given instant of time to find the fluid particle at a given position with a given velocity. The PDFs evolve according to the discrete Boltzmann equation. Macroscopic flow quantities (density and velocity) are obtained as zero and first order statistical moments of the PDFs. The difference between these macroscopic quantities and those that would be obtained by solv ing the Navier-Stokes equation is vanishing for small values of the Mach number (Ma < 1), i.e. for uncom pressible flows: in other words the LBM is said to be equivalent to the Navier-Stokes equation (Chen & Doolen 1998) for small Mach numbers. More recently,

it has been shown that such a limitation can be removed by adopting suitably complex sets of PDFs (Sun & Hsu 2003). The LBM has been applied also to the simulation of the advection-diffusion of contaminants (Shan 1997), revealing its potentialities as model for various physical phenomena.An interesting case of advectiondiffusion of contaminant is represented by a negatively buoyant dense jet entering a crossflow of light ambient fluid. Practical aspects of this seemingly theoretical problem are well addressed in Lai & Lee (2014): they are connected mainly to the release of brine in water bodies as waste product of desalination processes. Indeed, as for domestic wastewater discharges, the brine is typically released from a submarine outfall as a dense jet in a lighter environment in order to maximize the dilution. The problem has been investigated both experimentally, as e.g. in Papanicolau et al. (2008) and Papanicolau et al. (2008) and Lai & Lee (2014), and numerically, as in Wang et al. (2011), Decrop et al. (2015) and Craske & van Reeuwijk (2015). Numerical approaches consist mainly in Direct Numerical Simulation (DNS) or Large Eddy Simulation (LES) of the Navier Stokes equation. Very recently a LBM-based DNS of a round jet in crossflow (Lei et al., 2015) has been performed. However in this case the density of the jet and that of the ambient fluid is the same. In this work, we want to perform a first assessment on the ability of the two dimensional LBM in simulating a negative buoyant saline jet entering an uniform

Figure 1. Sketch of the 2DQ9 lattice.

flow. The difference between the jet density ρ_j and the ambient fluid density ρ_0 is small, so that the Boussinesque's approximation can be be assumed safely.As a consequence the density variations affect the flow only by means of the gravity force term and the incompressibility hypothesis is kept. Mixing of salt is accounted for. Numerical results are compared by experimental ones, conducted at the Hydraulics Laboratory of the Bari Politecnico. Experimental and numerical density contours and velocity plots are compared. A qualitative good agreement is found, showing that the LBM is a valuable tool in simulating such flows. Further work is to be done in order to perform fully 3D numerical simulations.

2 THE LBM FOR ADVECTION-DIFFUSION OF CONTAMINANT

The lattice shown in figure 1 is considered at each point P≡ (x, y) of the two dimensional fluid domain D. The arrows refer to the directions along which the fluid particle is allowed to move, with a given parti cle velocity c_i, $(i = 1, \ldots, 8)$. Define $f_i(x, y, t)$ the PDF

relative to a fluid particle traveling with velocity c_i.

The latter evolves according to the discrete Boltzmann

equations:

for $i = 0, 1, \ldots, 8$. τ^* is the relaxation time, f^{eq}_i the

equilibrium PDF relative to the ith lattice direction,

F_{ext} the external force. N_c is an integer, which for

the 2DQ9 lattice is equal to 6 and c_0 is the modu

lus of the fluid particle velocity. Adopt a space-time

discretization with constant space and time interval

s, �t. Firstly, it is usual to define c_0 as the ratio of �s

on �t. Secondly, adopting a lagrangian discretization for left hand side of equation 1, the LBM algorithm is obtained: The ratio τ^*/�t is the dimensionless relaxation time τ, while c_{xi}, c_{yi} are the cartesian components of the particle velocity vector c_i. In this work the external force F_{ext} is the gravity force: where ρ_s is the density of the salt and j the unit vector aligned with the vertical direction, upwardly oriented. $C = C(x, y, t)$ is the salt concentration, measured in m^3 of salt per m^3 of ambient fluid. Finally, the equilibrium PDF relative to the ith lattice direction f^{eq}_i is defined as: The weighting coefficients w_i are defined as: Flow density ρ_0 and velocity u are obtained by means of: Definition 4 is an approximation of the maxwellian distribution function. Equations 1 (and then its discretised counterpart 2), together with the definition of the external force 3, of the equilibrium PDF 4 and of the macroscopic quantities 6 are equivalent to the mass conservation and the Navier-Stokes equation, with the Boussinesque forcing term: where ν^* is a characteristic LBM viscosity, defined as:

For highly turbulent flows, such as the ones object of

this study, the resort to turbulence closure models is mandatory in order to keep computational costs compatible with ordinary computational resources. In this study a Large Eddy Simulation (LES) closure based on the Smagorinsky formulation of subgrid turbulence stresses is employed (Hou et al., 1994). The evolution of the concentration C is obtained in the framewotk of the LBM by introducing the PDFs for the concentration χ_i. They behave similarly to the f_i, i.e. their evolution is determined by the algorithm:

for $i = 0, 1, \ldots, 8$. The meaning of the various symbols is straighforward. $\tau_c = \tau^*_c /$▮t is the dimensionless relaxation time for the concentration, while χ^{eq}_i is the equilibrium PDF for the concentration, relative to the ith direction of the lattice. The expression of χ^{eq}_i is identical to 4, with C instead of ρ_0. The concentration C is then obtained as:

Equation 9, together with the definition of the equilibrium PDFs and the equation 22 is equivalent to the advection diffusion equation for the concentration:

where κ^* is a characteristic LBM diffusion coefficient, defined as:

Once C is calculated from 22, the actual density of the flow is defined as:

The numerical calculation of the density ρ_0, the velocity field u and the concentration C, based on the

LBM, is intrinsically dissipative, the LBM viscosity and diffusivity coefficients being defined by 8 and 12 respectively.

3 COMPUTATIONAL CONSIDERATIONS

Numerical simulations have been performed on a 2D, rectangular domain, $L \times H$, with $L = 0.5$m and $H = 0.28$m (H is the channel height, see Table 1). The fluid domain, placed along the center line of the channel (figure 2), has been discretised with 2000 cells along x and 1120 cells along y. At time $t = 0$ the density has been set to the ambient density value ρ_0, while the velocity field has been set to a constant value u, equal to the crossflow velocity. The initial concentra

tion has been set to 0 everywhere. The initial values of the PDFs have been set to those assumed by the corresponding equilibrium PDFs. On the bottom of the domain, at $y = 0$, except for the jet inlet, the secondorder bounce-back rule (Succi 2001) has been applied to impose a no-slip boundary condition. On the jet inlet and the left side of the domain, at $x = 0$, a constant velocity field is imposed according to Bouzidi et al. (2001). On the left side of the domain, at $x = L$ a zero gradient condition (Succi 2001). On the top of the domain, at $y = L$ the second-order bounce-forward rule (Succi 2001) has been applied to impose a freeslip boundary condition. From a computational point of view, the dimensionless parameters governing the considered phenomenon are: the jet Reynolds number Re, the jet Grashof number Gr and the Prandtl number Pr. In terms of physical quantities and with reference to the considered phenomenon they are defined as: where d, is the diameter of the jet inlet, U is the inlet jet velocity, ν, κ are the fluid kinematic viscosity and the diffusivity coefficient respectively, C_0 is a characteristic concentration (e.g. the concentration at the inlet of the jet). The parameters 14 can be also defined in terms of Lattice Boltzman Units (LBU). The latter arise from a particular scaling of the LBM, which consists simply in choosing ∆s = 1, ∆t = 1. As a consquence $c_0 = 1$. This fact permits to perform the LBM numerical calculation in similarity conditions with the considered experimental case. Indeed we start choosing the values of τ, τ_c, on the basis of stability considerations ($\tau > 0.5$, $\tau_c > 0.5$; Succi, 2001). Then the LBM kinematic viscosity and diffusivity coefficient can be calculated by means of 8, 12, adopting the LBU scaling: The values of the Reynolds, Grashof and Prandtl numbers, expressed in LBU, have to coincide with the values of the corresponding numbers, expressed in terms of physical quantities:

Figure 2. Definition sketch of the rectangular flume with crossflow and the circular feed tank of the salt solution.

where d_{LBU}, U_{LBU} and g_{LBU} correspond to the quantities d, U, g in 14, but are expressed in LBU. From the first two equations 16, it is possible to determine two of the three quantities d_{LBU}, U_{LBU}, g_{LBU}, after having set one of them. The (desired) consequence is the fulfilment of the similarity conditions. It is interesting to observe that the expression of the Prandtl number in LBU is given by:

Definition 17 permits to avoid arbitrary definition of τ, τ_c. In this work the Prandtl number is assumed equal to 1, thus applying definition 17, the same values are obtained for τ, τ_c.

4 EXPERIMENTAL SETUP

The experimental setup is placed at the Laboratory of Coastal Engineering of the DICATECh of the Technical University of Bari, Italy, and consists of a rectangular channel (figure 2), length 15m, width

4m, depth 0.40m, whose base and lateral walls are made of transparent glass. Experimental conditions are resumed inTable 1.The ambient crossflow is generated by a recirculating flow system. The channel flow rate is supplied from a downstream metallic tank by a Flygt centrifugal electro-pump and then discharged into the upstream steel tank, where a side-channel spillway with adjustable height is fitted and a conductivity probe measures the salinity. The water that overflows is directed into a parallel pipe and finally discharged into the downstream tank, while the required flow rate is released into the channel and measured by electro magnetic flow meters. The flume is equipped with a sliding support for measurement instruments, which enables them to displace along the three spatial direc tions x, y, z. A vertical nozzle (diameter d) is placed exactly at the center of the flume, at x = 0, y = 0, z = 0. A known weight of table salt (NaCl) is added to fresh tap water in a calibrated, circular storage tank. To keep constant the salinity, the whole volume of salty water

Figure 3. Sketch of the jet entering the crossflow.

is kept mixed by means of compressed air jets. A conductivity/temperature probe is mounted in this tank to measure the initial jet salinity. The salty water is pumped to the jet nozzle and a magnetic flow meter measures the flow rate Q_j, which is kept constant by sending back the excess brackish water to the feed tank. The salinity fields are measured at several longitudinal and transversal sections by means of a Micro Scale ConductivityTemperature Instrument (MSCTI), designed to measure the temperature and electrical conductivity of water solutions containing conductive ions. The MSCTI provides analog voltage outputs that are functions of the solution's electrical conductivity and temperature. The linear conductivity extends from $0.05 Sm^{-1}$ to $80 Sm^{-1}$, while the output voltage range is ±5V . The MSCTI gives accurate results for moving fluids too. The instantaneous three-dimensional flow velocity components are measured using the Vectrino Acoustic Doppler Velocimeter (ADV) system, together with the CollectV software for data acquisition and ExploreV software for data analysis. The basis measurement technology is the coherent Doppler processing, characterized by accurate results with no appreciable zero offset. Vectrino is used with a velocity range of $\pm 0.30 ms^{-1}$, a velocity accuracy of ±1%, a sampling rate of 100Hz, a sampling volume of vertical extension of 5×10^{-3} m and a time of acquisition of 60s. Both Vectrino and MSCTI are mounted on the sliding support. Measurements are carried out at various distances along the streamwise direction: from x =−1d to x = 100d, and along the vertical direction from z = 0, at the jet exit's center, to z = 60d. Extensive measurements of flow salinity and velocity are taken in the plane of flow symmetry (y/d = 0). 5 RESULTS AND DISCUSSION As the jet enters the ambient crossflow, it starts mixing with the fluid ambient and rises until it reaches a maximum height Z_{max} , then it starts falling and finally spreads along the bottom as a gravity current (figure 3). We consider two of the most meaningful quantities (Gungor & Roberts 2009): the maximum height Z_{max} and the dilution S of the jet, i.e. the ratio of the jet discharge Q at a given position x, y, z on the jet discharge Q_j at the inlet (figure 3). These quantities Table 1. Experimental conditions realised at the Hydraulics Laboratory of the Bari Polytechnical University. Case 1 2 Channel height H 0.28m 0.28m Crossflow discharge q $0.0448 m^3 s^{-1}$ $0.0448 m^3 s^{-1}$ Crossflow velocity u $0.04 ms^{-1}$ $0.04 ms^{-1}$ Density ρ_0 $998.5 kgm^{-3}$ $996.8 kgm^{-3}$ Kinematic viscosity ν $1 \times 10^{-6} m^2 s^{-1}$ $1 \times 10^{-6} m^2 s^{-1}$ Crossflow Reynolds n. Re_c 1.1×10^4 1.1×10^4 Jet diameter d 5×10^{-3} m 1×10^{-2} m Jet discharge Q_j $1.7 \times 10^{-5} m^3 s^{-1}$ $1.7 \times 10^{-5} m^3 s^{-1}$ Jet velocity U_j $0.85 ms^{-1}$ $0.21 ms^{-1}$ Jet density ρ $1010.2 kgm^{-3}$ $1007.2 kgm^{-3}$ Jet kinematic viscosity ν $1 \times 10^{-6} m^2 s^{-1}$ $1 \times 10^{-6} m^2 s^{-1}$ Jet Reynolds number Re 4×10^3 2×10^3 Jet Grashof number Gr 1.2×10^4 6.8×10^4 Table 2. Experimental, numerical and predicted values of Z_{max} , S_t , S_i , x_i . Lai & Lee Gungor & Roberts Case Quantity Experimental Numerical (2014) (2009) 1 Z_{max} (m) 0.25 0.21 0.23 0.36 1 S_t 20 20 39 39 1 S_i 97 97 2 Z_{max} (m) 0.11 0.10 0.10 0.15 2 S_t 6 5 6 6 2 S_i 11 10 16 16

can be expressed as power laws in terms of the dimen sionless parameter $U_r Fr$, product of the jet densimetric Froude number Fr, defined as:

and the ratio U_r is of the ambient crossflow velocity u on the inlet jet velocity U_j. The dimensionless parameter $U_r Fr$ is practically a densimetric Froude number of the ambient crossflow. According to Lai & Lee (2014), for $0 < U_r Fr < 0.8$ the ambient flow is considered weak, almost stagnant, and the jet dominates, while for $U_r Fr > 1$ the ambient crossflow dominates on the jet, affecting significantly its behaviour. Under the action of the ambient crossflow the jets bends over. For increasing values of $U_r Fr$ the bending of the jet increases and the jet's maximum height decreases. The power law of the maximum height Z_{max} is given by:

The values of the constant K and the exponent n depend on the considered regime: in the dominating jet regime ($0 < U_r Fr < 0.8$), according to Gungor & Roberts (2009) and Lai & Lee (2014), $n = 0$, $K = 2 \div 3$. In the crossflow dominated regime ($U_r Fr > 1$) Gungor & Roberts (2009) propose $n = 1/3$, $K = 2.5$, while Lai & Lee (2014) and previous authors (Lindberg 1994) agree rather on $n = 1/2$, $K = 1.8$. The dilution S of the jet is considered both at the maximum height of the jet's centerline (figure 3) where $S = S_t$, and at $z = 0$, $x = x_i$, where $S = S_i$ (figure 3). In both cases the power law assumes the expression (Gungor & Roberts, 2009; Lai & Lee, 2014): The constant K_S is equal to $0.8 \div 0.9$ for S_t and to 2 for S_i. The jets considered in this work belong to the crossflow dominated regime. Indeed the values of the parameters $U_r Fr$ are equal to 1.8 and 1.5 respectively for case 1 and 2. Table 2 resumes the values of Z_{max}, S_t, S_i for case 1 and 2. The numerical value of the dilution S has been determined considering that the contaminant's mass discharge is constant along the centerline of the steady jet: From equation 21 follows that:

Figure 4. Contour plots of the concentration. Numerical results (a) and experimental data (b) of Case 1. Numerical (c) and

experimental (d) results of Case 2.

being C_j the concentration of contaminant within the

jet at the inlet.The meaning of the dilution is how many

times the concentration C_j at the inlet is greater than the

concentration C at a given position on the centerline of

the jet. The agreement between numerical and exper

imental values is fairly good for Z_{max}. The value of

Z_{max} predicted by means of the power law of Gungor & Roberts (2009) is rather overestimated, while the formula of Lai & Lee (2014) gives values in reasonable agreement with both numerical and experimental values. The agreement on the dilution values between numerical and experimental values is quite good, while predicted values are generally larger than numerical and experimental ones. Numerical and experimental

contour plots of the concentration are shown in fig

ures 4a, 4b and in figures 4c, 4d, respectively for

cases 1, 2. The agreement between numerical and

experimental contour plots for both cases is fairly

good from a qualitative point of view: the numeri

cal shape of the jet coincides to a considerable extent

with its experimental counterpart. From a quantitative

point of view there are evident differences between the

experimental and numerical contour plots shown in in figures 4a, 4b and in figures 4c, 4d. This fact depends on the three-dimensional effects of the considered phe nomenon, which become increasingly important along the development of the experimental jet. Although the latter is initially axisymmetric, the ratio of the size of the channel to the nozzle's diameter is so large that the two-dimensional numerical model is able to reproduce fairly well the jet's behaviour in its ini tial development. Three-dimensional effects, caused by the ambient crossflow, become more and more important following the jet's centerline and they cannot be accounted for by the two-dimensional numeri cal model. For this reason differences appear clearly between numerical and experimental concentration's values at the end of the considered fluid domain. It should be emphasized that in figure 4d the concen tration of the ambient cross flow far from the mixing zone of the jet is different from zero, i.e. there is a sort of residual concentration field, because the hydraulic circuit of the channel is closed. However, thanks to the size of the channel, much larger than the diameter of the jet's nozzle, the residual concentration field of the ambient fluid remains very low during the experiments and does not affect the measured concentration values within the jet.

6 CONCLUDING REMARKS

In this work a negatively buoyant jet in an ambient crossflow has been numerically simulated by means of the Lattice Boltzmann Method. The negatively buoyant jet is a solution of water and salt (NaCl) with given concentration, while the ambient cross flow is fresh water. The considered phenomenon is a model for the release of brackish water by desali nation or energy production plants based on reverse osmosis. The Lattice Boltzmann model is equivalent to the incompressible Navier Stokes equation, with a gravitational forcing term defined according to the Boussinesque hypothesis, and to the advection dif fusion equation for the concentration. The numerical simulations have been realised on a two-dimensional grid belonging to the vertical plane along the centerline of the channel. Experiments, realised at the LIC – Lab

Interfacial instabilities of gravity currents in the presence of surface waves

L.M. Stancanelli, R.E. Musumeci & E. Foti

Department of Civil Engineering and Architecture, University of Catania, Catania, Italy

ABSTRACT: The salt-brackish wedge propagation in the presence of regular surface waves is investigated by

adopting a lock-exchange scheme. The aim is to investigate the dynamics of the fore front and of the instabilities

at the interface.An experimental campaign has been carried out by considering different values of reduced gravity

(range of g' 0.04 ÷ 0.15 m/s ^{2}) and different wave conditions (i.e. wave height range 1.5 ÷4.3 cm, wave period range 0.7 ÷ 1.3 s). Experimental data confirm that the presence of the waves significantly modifies the dynamics of the front density by inducing a reduction/increase of the average velocity of the front. The instabilities at the interface have been analyzed both experimentally and by means of CFD numerical simulations. In particular, thanks to the calibration with the experimental data, the numerical simulation is able to provide information on the turbulence formation and mixing process at the interface. Results confirm that in the presence of waves the mixing at the interface is importantly influenced by the particle displacement due to the oscillatory motion.

1 INTRODUCTION

Gravity currents are also known as density currents or buoyancy-driven currents. They occur when a fluid flows in a fluid with a different density. Such currents are quite common both in natural and man-made systems and their study has many applications in the field of hydraulic and environmental engineering. Saline intrusion, outflows of industrial plants, oil spills in the ocean, turbid water intrusion in a lake, suspended sediment plume in a river or in coastal environment and just some of the most relevant examples. The field investigation of gravity currents is usually very difficult due to their complicated and unexpected characteristics (An et al. 2012). Laboratory experiments have been often adopted to investigate the dynamics of such a phenomenon. In particular, in the past, the lock exchange scheme was extensively adopted considering different configurations (Benjamin 1968,Turner 1973, Ungarish 2009) such as: full-depth and partial depth two-dimensional cases (Huppert and Simpson 1980, Shin et al. 2004, Lombardi et al. 2015), axisymmet ric cases (Hallworth et al. 2001, Ungarish and Zemach 2007) or three-dimensional gravity currents over both smooth and rough bottoms (La Rocca et al. 2008, Nogueira et al. 2013, Nogueira et al. 2014). Also numerical studies on the gravity currents have been carried out. Direct Numerical Simulations (DNS), car ried out for investigating lock-exchange flow in an infinitely long channel, were able to provide a reason able and highly accurate description of the lobe and cleft instability at the head of the front (Härtel et al. 2000). Large eddy simulation (LES) performed by Ooi et al. (2007) reproduced the dynamics of the gravity current propagation considering both qualitative and quantitative aspects. Good reproduction of the velocity propagation and of the dynamics of the instabilities at the interface has been obtained when compared with the DNS results of Härtel et al. (2000). An et al. (2012) investigated saline intrusion, by applying the FLOW-3D model and by evaluating the performance of such numerical model. Two different turbulence closure schemes have been considered, namely the Renormalization Group (RNG) $k-\varepsilon$ scheme in a Reynolds-averaged Navier Stokes framework

(RANS) and the Large Eddy Simulation (LES) technique using the Smagorinsky scheme. FLOW-3D results allowed to predict the temporal and the spatial evolution of the intrusive gravity currents. Moreover, discrepancies between RNG $k - \varepsilon$ and LES closure schemes were negligible in terms of the velocity of propagation of the gravity current. Notwithstanding the fact that the discharge of fresh or brackish water in the sea is frequent, the effect of the wave motion on the propagation of the salt-brackish wedge in coastal regions has received much less attention (Robinson et al. 2013). The aim of the present work is to investigate the influence of regular surface waves on the dynamics of Boussinesq gravity currents. In particular, specific objectives are the assessment of the change of the front velocity and the study of the mixing process at the interface in the presence of surface waves. In order to cope with these problems both an experimental and a numerical approach have been adopted. In both cases the lock exchange schematization has been used. In particular, the numerical calculations reproduced the same conditions of the experimental set up. The commercial CFD model FLOW-3D is selected as numerical model, since it has been previously extensively tested and validated (Yang et al. 2010, Ozmen-Cagatay and Kocaman 2011, Erduran et al. 2012). The paper is divided in two main sections. The first one reports a detailed description of the experimental setup, the list of experiments carried out and a discussion of the experimental results. The second section presents a brief description of the numerical model adopted, the definition of the boundary conditions and of the computational domain along with a discussion of the results of the numerical simulations.

2 PHYSICAL MODELING

2.1 Experimental apparatus and experiments

The experiments are carried out at the small-scale wave channel of the Hydraulic Laboratory of the University of Catania.The flume is 9 m long, 0.5 m wide and 0.7 m high. It is equipped with a piston-type wavemaker and it has been partitioned at its central section by means of a Perspex sluice gate, in order to execute classical lock exchange tests. At the end of the flume a porous beach allows to minimize wave reflection. Salt water, having density ρ_1, is present at the wavemaker side of the gate, and fresh water, having density $\rho_0 < \rho_1$, at the onshore side. Household salt and green food dye are used in all the experiments to generate and to highlight the density difference between the two fluids. Full-depth two-dimensional lock-exchange experiments have been carried out with and without regular waves. Figure 1 shows the experimental apparatus previously described. An automatic optical measurement methodology has been implemented to detect the geometric and kinematic characteristics of the front propagation, such as: depth, velocity and interface. In particular, the measuring area is about 1.6 m long in the direction of the current propagation.The kinematic characteristics of the wave motion (i.e. wave height, wave period) have been measured by means of acoustic sensors. Table 1 reports the controlling parameters of the experiments, i.e.: H still water level within the tank, ρ_1 salt water density, ρ_0 fresh water density, g' reduced gravity $g' = g(\rho_1 - \rho_0)/\rho_0$, H_w wave height and T_w wave period. Different reduced gravity values (range of g' 0.039 ÷ 0.145 m/s^2) and different wave conditions (i.e. wave height range 1.5 ÷ 4.3 cm, wave period range 0.7 ÷ 1.3 s) have been considered.

2.2 Experimental observation

Experimental evidences obtained in the absence and in the presence of the wave motion are presented here. All experimental results refer to gravity current propa gation during slumping phase. The density difference is always such that is satisfies the Boussinesq approx imation ($\rho_1/\rho_0 \sim 1$). Gravity currents in still ambient fluid are reproduced considering three different val ues of reduced gravity (W101 g' = 0.14 m/s^2; W111 g' = 0.04 m/s^2; W115 g' = 0.08 m/s^2). Present experi mental results agree with classical scientific literature. Indeed, constant values of the average front velocity are observed, which agree with to the solution pre sented by Huppert & Simpson (1980) for the flow

Table 1. Controlling parameters of the experimental campaign.

Run	H [cm]	ρ_1 [kg/m^3]	ρ_0 [kg/m^3]	g' [m/s^2]	H_w [cm]	T_w [s]
W101	20.3	1011.4	997.2	0.14	–	–
W102	20.3	1011.7	997.0	0.15	1.5	1.3
W103	20.3	1010.6	997.0	0.13	2.5	1.0
W104	20.0	1010.3	997.3	0.13	2.9	0.8
W105	20.1	1010.4	997.1	0.13	3.9	0.7
W106	23.0	1000.9	997.2	0.04	1.8	1.3
W107	20.3	1001.5	997.5	0.04	2.5	1.0
W108	20.0	1002.0	996.2	0.06	3.1	0.8
W109	20.1	1001.8	997.8	0.04	4.3	0.7
W110	20.3	1003.2	997.5	0.06	1.6	1.3
W111	20.3	1001.7	997.8	0.04	–	–
W112	20.3	1005.9	997.5	0.08	1.6	1.3
W113	20.3	1006.6	997.0	0.09	2.6	1.0
W114	20.0	1006.1	997.1	0.09	3.0	0.8
W115	20.3	1006.0	997.8	0.08	–	–
W116	20.1	1006.9	997.3	0.10	4.2	0.7
W117	20.0	1006.7	997.8	0.09	0.5	3.0
W118	20.2	1006.9	997.6	0.09	0.7	1.8
W119	20.4	1006.8	998.0	0.09	1.1	1.5
W120	20.0	1006.4	997.9	0.08		

2.1 0.7 W121 20.0 1006.5 997.8 0.09 1.2 1.0 W122 20.0 1006.5 997.6 0.09 3.4 1.0 W123 20.2 1001.6 997.7 0.04 0.8 1.8 W124 20.4 1001.2 997.9 0.03 1.2 1.5 W125 20.2 1011.2 998.1 0.13 0.8 1.8 W126 20.4 1011.8 997.6 0.14 1.0 1.5

slumping phase. As expected an increase of reduced gravity corresponds to higher speed of the front propagation v_f (W101 v_f = 0.65 m/s; W111 v_f = 0.50 m/s; W115 v_f = 0.60 m/s). Moreover, different shapes of the front are observed for different values of reduced gravity. In particular, as it is possible to observe in Figure 2, a less steep head front is observed for higher value of reduced gravity (slope range is 30°–60°). Moreover, larger reduced gravity are responsible of more intense development of turbulence structures (see Figure 2). Indeed, in the present experiments, the larger Grashof numbers: are respectively: the $3.3 \cdot 10^4$, $6.7 \cdot 10^4$, $1.2 \cdot 10^5$). Gravity current propagation is significantly influenced by the superimposition of the orbital motion. Regular surface waves are generated in intermediate water depth conditions. Orbital velocities are function of the wave phase and of the position of the fluid particle along the water column (see Figure 3).Therefore in the presence of regular waves, the gravity current is governed by two different driving forces: buoyancy and the inertial force due to the oscillating motion. Analyses of the front propagation are carried out throughout the wave period. Without waves the velocity of propagation is constant (slumping stage), while in the presence

Figure 1. Experimental apparatus for lock-exchange experiments in the presence of regular waves.

Figure 2. Gravity current evolution in case of different

reduced gravity g': 0.04 m/s^2; 0.08 m/s^2; 0.14 m/s^2.

of regular waves a systematic modulation of the front

shape and of the propagation velocity is observed (see

Figure 4 and Figure 5). In particular, when the grav

ity current is influenced by the passage of the crest

the propagation velocity increases. An intermediate

propagation velocity is observed at the phase of zero-

up crossing and zero-down crossing. A reduction of velocity is observed as the wave trough passes (see Figure 4). It is also clear that also an oscillation of the interface along the core of the gravity current occurs at the zero-up crossing and zero-down crossing wave phases (see Figure 5). The vertical amplitude of such oscillations contributes to determine the minimum and maximum depth of the current. Moreover the ampli tude of the vertical oscillations at the interfaces is

Figure 3. Sketch of velocity field along the water column in the presence of regular waves. Figure 4. Gravity current propagation during a wave period: at g' 0.10 m/s 2 , H_w = 3.35 cm, T_w = 1.02 s. similar to the elliptical particle displacement in intermediate water depth. In particular, the amplitude of the vertical displacement B of a particle according to the linear theory can be calculated as (Dalrymple and Dean 1991): where H_w is the wave height, H is the water depth, k is the wave number, σ is the angular frequency of

Figure 5. Gravity current interface during a wave period: at g' 0.10 m/s 2 H_w = 3.35 cm T_w = 1.02 s L = 1.26 m.

the waves and z_1 is the elevation. Finally a compar ison between experiments, characterized by similar buoyancy driving force and carried out in the absence and in the presence of wave motion, shows that large instabilities are damped in the presence of the regu lar waves (Figure 7). Figure 6 shows the oscillation of the full front interface and the change of shape of the interface according to the wave phase. A red refer ence line is plotted in order to more easily recognize such a phenomenon. More in details, when waves are superimposed to the gravity current, the development of K-H billows and of small scale turbulence structures tends to be inhibited.

3 CFD MODELING

3.1 Model description

FLOW-3D has a variety of turbulence models for simulating turbulent flows, including Prandtl mixing length model, one-equation model and two-equation $k - \varepsilon$ model, Re-Normalisation Group (RNG) scheme, and Large Eddy Simulation LES model. These turbulence models have been well tested and documented in the relevant technical literature (Ooi et al. 2007). Based upon prior experience with FLOW-3D, the RNG $k - \varepsilon$ model was selected for all simulations since it is able to catch the main characteristics of the flow at a relatively low computational cost. FLOW-3D employs finite volume difference method to discretize the computational domain. In particular, the physical domain to be simulated is decomposed by using Cartesian grids composed of variable size hexahedral cells. The continuity equation and momentum equations are solved

along with the turbulent closure $k - \varepsilon$ equations: Figure 6. Gravity current core depth oscillation caused by the wave motion, when a wave with period of 0.71s is superimposed. where k = turbulent kinetic energy, ε = turbulent energy dissipation rate, G = rate of production of turbulent kinetic energy, i = 1, 2, 3, j = 1, 2, 3, δ_{ij} = Kronecker delta function, ν = kinematic viscosity, ν_t = eddy viscosity, u_i velocity component in the ith direction, p =

pressure, ρ = fluid density, g_i = gravity acceleration component in the ith direction ($g_1 = g_2 = 0$ and $g_3 = -g$). The coefficients in Eqs. (5) and (6) are chosen based on the classical model by Launder & Spalding (1974) as C_d = 0.09 ($C_{1\varepsilon}$ = 1.44, $C_{2\varepsilon}$ = 1.92, σ_k = 1.00, σ_ε = 1.30). 3.2 Numerical simulations Simulations are performed under conditions that correspond to the laboratory experimental setup described in section 2.1. The computational domain similar to the experimental dimensions (i.e. channel length 9 m, channel width 0.5 m and channel elevation 0.7 m). The numerical model deployed a regular grid system of $9.3 \cdot 10^5$ cubic cells (size 0.015 m). The time step was set equal to 0.01 s and the total simulated time for each run was about 15 s requiring a computational

Figure 7. Gravity current propagation in the case of gravity current characterized by g' of about 0.090 m/s^2: a) without

waves; b) with waves. The top panels represent the images as acquired by the video recording system. The bottom panels

represent black and white images of the front, where the shape of the interface is easily recognizable.

Figure 8. Computational domain and boundary conditions

selected for the CFD simulation of gravity currents.

time of about 9 min. All surfaces of the flow domain

were defined as no-slip smooth walls, except the free

surface where a constant pressure is selected as bound

ary condition. The different fluids are contained in

two initially different mesh blocks. A zero gradient

boundary condition is used at the interface of the two

fluid mesh blocks. The computational domain depict

ing the adopted meshes and boundary conditions is

illustrated in Figure 8. For the case of gravity current

in the presence of waves at the offshore end of the saltier side a regular wave field is generated and it enters the domain. Moreover in such a case, in order to avoid wave reflection from the onshore boundary, the channel length of the ambient fluid is 30 m long, so that both the current and the waves never reach the end of the flume within the simulation time. The CFD model was applied to two experimental tests. In case 1, the model was applied to a classical lock exchange problem, reproducing experiment W101. In case 2, the model was applied to a lock exchange release but in the presence of regular waves, reproducing experiment W104. The 3D model results were firstly compared with experimental observations in order to calibrate the numerical model. Once calibrated, CFD numeri cal results allow a more detailed understanding of the phenomena, such as the mixing at the interface and the 3D vorticity dynamics, which are difficult to be measured in the lab. Figure 9 shows results for case 1 at the initial stage of the model simulation. A steep front is observed caused by the lock release. After that, the intrusion of the saltier fluid in the ambient fluid is clearly recognizable, a constant propagation veloc ity of 0.65 m/s is observed (equal to one measured in the lab). A steep front propagate forward and appear ance of turbulent structures on the core interface are observed. Due to the low Reynolds number (6000) that Figure 9. CFD simulation of gravity current $g' = 0.10$ m/s 2 and in the absence of surface regular waves (from the top to the bottom) at t = 0 s, t = 1.2 s, t = 2.2 s; t = 4.6 s. characterizes the tests, no Kelvin–Helmholtz instabilities are visible during the simulation, in accordance to the experimental observations. Finally, in the case 1 a mixing layer can be noticed, which is characterized by a thickness of 1–2 cm, comparable to the dimension of the fluid of lighter green color acquired during the lab video recording. In case 2, the gravity current propagation in the presence of waves is reproduced and the influence of the wave motion is investigated. Figure 10 shows a sequence of snapshots along a wave period (0.84 s). Differences on the instantaneous front velocity are observed at different wave phases, which result in a different displacement of the front. In particular for the investigated case a decrease of the average front speed is observed. The oscillation of the core and the consequent change of the gravity current depth are also reproduced. Such phenomena are clearer if a video of the front propagation is observed.

Figure 10. CFD simulation of gravity current $g' = 0.10$ m/s 2 and in the presence of surface regular waves ($H_w = 2.86$ m

$T_w = 0.84$ s L = 0.95 m). The depth variation at the gravity current interface

in space and in time is also observed. Moreover, the

model fairly predicted the macroscopic density varia

tion of the measurements. Finally, the damping of the

instabilities structures at the interface is clearly recog

nizable also in the numerical simulations as it was in

the experimental data.

4 CONCLUSIONS

The analysis of gravity currents has been carried out by

means of full depth lock-exchange experiments both

in the absence and in the presence of regular sur face waves. Experimental and numerical approaches have been adopted for the investigation of the gravity current dynamics. Experiments have been performed varying the range of reduced gravity (high, medium and low reduced gravity) and the superimposed regular wave regime (4 different wave conditions). Prelimi nary analysis on experimental results in the absence of waves have been performed in order to validate the adopted measurement techniques and to use the results as a reference for the experiments carried out in the presence of waves. When analyzing the influence of regular waves on the gravity current dynamics, a strong interaction between the two flows has been observed and evaluated in terms of the dimension and of the velocity of the front of the heavier gravity current. The presence of the waves significantly modifies (up to 30%) the dynamics of the front propagation by induc ing a general reduction of the front velocity depending on the value of the reduced gravity. In particular, it causes: i) the dynamics of the flow becoming pulsat ing with the same period of the superimposed surface waves; ii) damping of the instabilities usually observed at the boundary between the denser and the lighter fluid. The numerical modeling carried out by means of the FLOW-3D model provided more information about the mixing layer dynamics and the influence of

the wave motion. In particular, the numerical simula

tion allows to increase the observational time, as in the

experimental model only a 1 m long measuring area

Huppert, H. E. & J. E. Simpson (1980). The slumping of gravity currents. Journal of Fluid Mechanics 99(04), 785–799.

La Rocca, M., C. Adduce, G. Sciortino, & A. B. Pinzon (2008). Experimental and numerical simulation of threedimensional gravity currents on smooth and rough bottom. Physics of Fluids (1994-present) 20(10), 106603.

Launder, B. E. & D. Spalding (1974). The numerical computation of turbulent flows. Computer methods in applied mechanics and engineering 3(2), 269–289.

Lombardi, V., C. Adduce, G. Sciortino, & M. La Rocca (2015). Gravity currents flowing upslope: Laboratory experiments and shallow-water simulations. Physics of Fluids (1994-present) 27(1), 016602.

Ng, C.-O. & S.-C. Fu (2002). On the propagation of a twodimensional viscous density current under surface waves. Physics of Fluids (1994-present) 14(3), 970–984.

Nogueira, H. I., C. Adduce, E. Alves, & M. J. Franca (2013). Analysis of lock-exchange gravity currents over smooth and rough beds. Journal of hydraulic Research 51(4), 417–431.

Nogueira, H. I., C. Adduce, E. Alves, & M. J. Franca (2014). Dynamics of the head of gravity currents. Environmental Fluid Mechanics 14(2), 519–540. Ooi, S. K., G. Constantinescu, & L. Weber (2007). A numerical study of intrusive compositional gravity currents. Physics of Fluids (1994-present) 19(7), 076602. Ozmen-Cagatay, H. & S. Kocaman (2011). Dam-break flow in the presence of obstacle: experiment and cfd simulation. Engineering applications of computational fluid mechanics 5(4), 541–552. Robinson, T., I. Eames, & R. Simons (2013). Dense gravity currents moving beneath progressive free-surface water waves. Journal of Fluid Mechanics 725, 588–610. Shin, J., S. Dalziel, & P. Linden (2004). Gravity currents produced by lock exchange. Journal of Fluid Mechanics 521, 1–34. Turner, J. S. (1973). Buoyancy Effects in Fluids. Cambridge University Press.

Cambridge Books Online. Ungarish, M. (2009). An introduction to gravity currents and intrusions. CRC Press. Ungarish, M. & T. Zemach (2007). On axisymmetric intrusive gravity currents in a stratified ambient-shallowwater theory and numerical results. European Journal of Mechanics-B/Fluids 26(2), 220–235. Yang, C., B. Lin, C. Jiang, & Y. Liu (2010). Predicting nearfield dam-break flow and impact force using a 3d model. Journal of Hydraulic Research 48(6), 784–792. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Unraveling salt fluxes: A tool to determine flux components and dispersion

rates from 3D models

W.M. Kranenburg, T. van der Kaaij, H. van den Boogaard & R.E. Uittenbogaard

Deltares, Delft, The Netherlands

Y.M. Dijkstra

Delft University of Technology, Delft, The Netherlands

ABSTRACT: This paper presents an instrument to determine salt flux components and equivalent dispersion

rates from 3D numerical model results for estuarine salinity. A spatial decomposition decomposes velocity

and salinity in cross-sectional averages and variations quantifying contributions of cross-sectional means and

variations to the total mean salt flux. A temporal decomposition separates the effect of storm-events and tidal

components by processing velocity and salinity signals with Godin filtering together with harmonic analysis.

From the spatial decomposition equivalent dispersion coefficients are computed, and applied in a 3D-1D numer

ical model comparison. This shows nearly complete reproduction. We conclude that the salt flux decomposition instrument provides valuable support in interpretation of 3D

model results by giving insight in the relative importance of various physical mechanisms, and in verification and improvement of 1D-models for quick assessments and derivation of statistics. The tool is available and in principle applicable in combination with any 3D-numerical model.

1 INTRODUCTION

In many delta's of the world salinity intrusion imposes limits to fresh water availability. Where globally the need for fresh water is increasing, at many places also salinity intrusion is expected to increase due to climate change through its effect on river discharge characteristics, sea levels and storm climates (WWAP 2015). In manipulated delta's like e.g. the Rhine-Meuse delta, salt intrusion is impacted by human activities as well. Salt intrusion is not only a relevant, but also a complex topic, due to the presence of a range of 3D mechanisms, and strong variations in space and time (Fischer et al. 1979, Geyer & McCready 2014). The relevance and complexity call for a proper system understanding (diagnosis), but also for means to predict and to explore salinity intrusion prevention measures based upon that (prognosis). For both system understanding and prediction of intrusion in scientific and engineering practice, 3D numerical models play an important role. However, interpretation of 3D model output in terms of pro cesses is complex: it is hard to see what is actually determining your results. Next, 3D models are often too heavy to carry out large numbers of computations, e.g. for derivation of salt intrusion statistics for future scenario's. For this type of work, often 1D models are used. However, with their limited process description and usual calibration on present or historic conditions, the value of this type of models for scenario's outside the calibration condition range or including changes within the system, is limited. In this paper, we present an instrument to determine salt flux components and equivalent dispersion rates from 3D numerical model results. The decomposition of salt fluxes, inspired by Fischer et al. (1979) and Lerczak et al. (2006), helps to understand the behavior of a physical system by giving insight in the relative importance of various physical mechanisms. The tool thus provides a simplification of the analysis of 3Dmodel results and provides possibility for diagnostic application. The determination of equivalent dispersion provides means to incorporate characteristics and qualities of 3D models into faster 1D models for quick assessments and derivation of statistics. In this way the tool expands the applicability of 1D-models in prognostic applications.This set-up is shown schematically in Figure 1. The salt flux decomposition tool uses both a spatial and a temporal decomposition. The spatial decomposition decomposes velocity and salinity in cross-sectional averages and variations to quantify the contribution of cross-sectional means and variations to the total mean salt flux. In the temporal decomposition we separate the effect of storm-events and tidal components by processing velocity and salinity signals with a Godin filtering in combination with harmonic analysis. From the spatial decomposition we compute dispersion coefficients, and apply these in a 3D-1D numerical model comparison for cross sectional averaged salinity. During the development, the various steps are tested for two cases: (1) an 3D numerical computation of an idealized trumpet-shaped estuary with M2 + M4-only forcing at the sea side and a constant discharge at the river side, and (2) a 3D numerical computation of an historic event of salin ity intrusion in the Hollandsche IJssel, a branch of the Dutch Rhine-Meuse delta system. Below, we discuss subsequently the spatial decom position (section 2), the temporal decomposition (sec tion 3), the computation of dispersion rates (section 4) and the 3D-1D comparison (section 5), including a selection of the tests. Section 6 and 7 give a discus sion respectively conclusions and outlook on further development of the decomposition tool.

2 SPATIAL DECOMPOSITION

2.1 Aim

In reality and in 3D-model results time-averaged and time-dependent velocity and salinity vary over the depth and the width of a cross-section. Next, these variations themselves vary along the estuary. The vari ations reflect the influence of processes like estuarine circulation, tidal straining (tide induced asymmetries in the density profile), tidal pumping (e.g. from mor phological influences), tidal trapping, and changes in the relative importance of these mechanisms along the estuary. The spatial decomposition aims to give insight in the relative importance of cross-sectional variations, and changes therein along the estuary. Next, it aims to quantify all the contributions that are not directly solved in a 1DH numerical model that only solves cross-sectional averaged time dependent flow velocity and salinity. All these contributions need to be accounted for in a 1D model through dispersion.

2.2 Method

Firstly we decompose along-estuary flow velocity u and salinity s in cross-sectional averages and varia tions. Next, we apply in this phase an averaging over n tidal cycles to get also a mean and variation over time.

Figure 1. Schematic overview of connections between mod els and components of this study. This results in the following four component decomposition (shown here for velocity u): u = u a + u b + u c + u d , u a = 〈 u〉 cross-sectionally-averaged time-averaged velocity u b = u − u a cross-sectionally-averaged time-varying velocity u c = 〈u〉 − u a cross-sectional variation of the time-averaged velocity u d = u − u a − u b − u c cross-sectional variation of the time-varying velocity u d = u − 〈u〉 − u − 〈 u〉 Here < > denotes averaging over time, and overbar averaging over the cross-section. When, next to this decomposition for velocity u and salinity s, we also include the crosssectional area with time averaged area A and time-variation a ′ , the total time averaged (i.e. residual) transport of salt can be decomposed in the terms: (see the report Kranenburg et al., 2016, for more details). From these terms, only four terms are a combination of merely cross-sectional averaged parameters. These terms are in principle solved by a 1D model with transport equation in the advection term: Term 1, which is a combination of time and crosssectional averaged velocity and salinity. Hereafter, it is referred to as ‘river’-transport because it could be seen as transport resulting from a background river current; and term 2, 5 and 6, which are the result of time variations of cross-sectional averaged velocity and salinity. Hereafter, we refer to the sum of these as cross-sectional averaged ‘tide’-transport. The remaining terms reflect influence of cross-sectional variations. In

1D-modelling, these terms should be accounted for in the dispersion term. The separate

terms all have there separate meaning, but for an over

all picture we now look to their sum, which we refer

to as ‘variation’-transport. In the present offline version of our salt

flux-decomposition instrument, all these terms are

computed from the output of 3D models with a mat

lab tool, available through https://svn.oss.deltares.nl/

re-pos/openearthtools/trunk/matlab/applications/sal_

decomposition_tool.

2.3 Test

Figure 2 and 3 show respectively the set-up and results

of a test for a trumpet-shaped estuary with variable

depth. Test case characteristics are: length 100 km,

width from 2500 m at mouth to 500 m at river side,

depth from 25 m at mouth to 5 at river side, with 2 m

lateral depth variation, forcing with M 2 water level

variation at sea side with amplitude of 2 m and a

constant discharge at river side of 100 m 3 /s, salinity

respectively 30 and 0.2 ppt, grid cells of 100 m by 1 2

the local estuary width horizontally, and 0.1 times the

Figure 2. Geometry and bathymetry of test case trumpet

estuary with variable depth.

Figure 3. Results for salinity fluxes for a test with a trumpet shaped estuary with variable depth. Top) grid 3D-model; Centre)

check for the total flux; Bottom) contributions of ‘river’, ‘tide’and ‘variation’-transport (negative is sea ward). X-axis: distance

from sea. depth vertically (i.e. 10 sigma layers of equal thickness). We carry out this simulation using the numerical modelling system Delft3D-Flow. Figure 3 shows that the time-averaged sum of the 16 individual contributions computed by our decomposition tool equals the time-averaged value of the salt flux computed directly from the model results using $\langle (A + a')us \rangle$. Next, the figure shows that the total net flux approaches the value of 20 kg/s that can be expected theoretically on the long term with the used boundary conditions. From this check we conclude that the terms are computed correctly. The figure also shows that the order of magnitude of the exporting river-transport and mostly importing tideand variation-transport can easily become two orders larger than the net transport, and that the relative importance of the various terms changes largely from mouth to river, with tideand variation-transport approaching to zero and river-transport approaching to −20 kg/s at the river side. These observations themselves are worth to be elaborated and investigated for other estuaries. But in the context of this paper they are especially illustration of the type of research that can be done with the salt flux decomposition, e.g. characterization of estuarine systems or research on the variation of importance of contributions with varying forcing conditions. 3 TEMPORAL DECOMPOSITION 3.1 Aim Figure 4 shows a time series of measured chloride concentration at Krimpen a/d IJssel for a period of a few months in 1998. From the figure various time

scales and temporal patterns can be recognized. These

scales and patterns encompass long term variations

induced by the discharge of the river Rhine, diurnal and

semi-diurnal tidally induced variations, wind induced

events, and/or other short term fluctuations.An impor

tant issue is then how salinity transport fluxes are

distributed over these various time scales and events.

Can we separate influence of tide and storm? And

what does this tells us about system characteristics?

Here we show a first attempt for such a temporal flux decomposition. We do this on the basis of a veloc ity and salinity time series for a period of about 15 days obtained from a numerical computation of salin ity intrusion in the Hollandsche IJssel. Compared to the series in Figure 4, the series used for this illus tration show more tide dominance and are much less complex because of the quasi-steadiness of the river discharge.

3.2 Method

The main step in our approach is a decomposition of the time series X_t (a velocity series u_t and/or a salinity series S_t) into three components:

Figure 4. Measured chloride concentration at Krimpen a/d IJssel, Feb-Aug 1998.

Figure 5. Results for time decomposition of a salinity signal (here model output for cross-sectionally averaged salinity at Krimpen a/d IJssel). Top) original signal (continuous line) and residual after Godin-filtering (dashed line); Bottom) Result of Godin-filter (dash-dotted black line), tide-related part of original signal (from hindcast, continuous line), and remaining residuals (dash-dot-dotted line). The $X^{(L)}_t$ is intended to be the component of X_t with the long(er) term variations, $X^{(A)}_t$ the component with tidally induced variations, and $X^{(R)}_t$ the remaining residuals representing all short(er) term fluctuations. The $X^{(L)}_t$ is derived with a low pass filter that removes, as much as possible, the tidal period variations in X_t . In practice

several filters are used for this removal. See for example Walters & Heston (1981) who consider the merits of a few alternatives commonly used in tidal analysis. Here we have adopted the Godin filter (Godin, 1972). In this filter a moving window average is applied three times, two times with a bandwidth of the 25 hours, and once with a bandwidth of 24 hours. Next, a harmonic analysis is applied to the ‘residuals’ ($X^{(L)}_t := X_t - X^{(L)}_t$) using the most important tidal constituents (in our example neglecting for the time being that the time length used there is actually too short for very precise distinction). The hindcast of this harmonic analysis serves as an estimate for $X^{(A)}_t$. Next $X^{(R)}_t$ is set to the remaining residuals: $X^{(R)}_t := X_t - X^{(L)}_t - X^{(A)}_t$. This decomposition is applied to both the velocity and the salinity time series. On this basis temporal averaged salinity fluxes can be computed for the various time scales C (as here denoted by L, A, or R) according to: For the tidal variations, encompassed in $X^{(A)}_t$, a refined decomposition is even possible, using the orthogonality of the involved harmonic constituents (indexed by k):

Table 1. Flux components: total net flux and contributions.

Component	Description/Source	Value
$\langle u_t \cdot S_t \rangle$	Total	-0.0633
(i) $\langle u^L_t \cdot S^L_t \rangle$	Events	-0.0835
(ii) $\langle u^A_t \cdot S^A_t \rangle$	Tide	0.0135
(iii) $\langle u^R_t \cdot S^R_t \rangle$	Residuals	0.0032
(i) + (ii) + (iii)	Sum	-0.0668

The A_k and k in this expression are the amplitude

and phase of the k th constituent.

3.3 Test

The results of this decomposition for the salinity time

series are graphically presented in Figure 5. The solid

curve in the upper panel shows the S_t generated within

the numerical computation while the dashed curve represents the series after removal of the non-tidal variations with the Godin filter. This signal, i.e. $S^{(L)}_t$, is shown in the lower panel (dashed curve), together with the identified tidal variations ($S^{(A)}_t$, solid curve) and the remaining residuals $S^{(R)}_t$ (dashed/dotted curve). In the plot of S_t clear non-tidal effects can be observed in the time interval from May 24 to May 29.This ‘anomaly’is properly recognized by the Godin filter, and exhibited by the two pronounced ‘bumps’ in $S^{(L)}_t$. Nevertheless the Godin filter does not fully succeed to remove the longer term variations. In fact, in the final residuals $S^{(R)}_t$ remaining effects of the events are notably found. This suggests to reconsider the Godin filter, and apply or develop alternatives for the removal of the non-tidal variations in the S_t. On the other hand it must be mentioned that in the first half of the period, the residuals $S^{(R)}_t$ are quite small in magnitude, and fluctuate on short time scales only. Table 1 and Figure 6 give a quantification of the found contributions to the salt fluxes.The sum of event, tide and residual contributions, with the tide contribution computed using equation 3, is not yet exactly identical to the total net flux, probably due to the limited record length, but is coming close. An interesting observation at this stage concerning processes is that the M2-contribution is rather small. This is probably

because flow and salinity are close to 90 degrees out

of phase in the Hollandsche IJssel.

4 EQUIVALENT DISPERSION

4.1 Aim

The aim of the next step is to derive dispersion coeffi

cients for 1D-models from the computed flux contri

butions. This is an important step from diagnostic to

prognostic application of the salt flux decomposition.
Figure 6. Contribution of various tidal components to net salt flux. Here we apply this step to the 'variation'-transport of section 2, the sum of all terms with cross-sectional variation together. 4.2 Method In 1D-models, the rate of change of salinity or chlorinity is described by an advection-dispersion equation like: with A the cross-sectional area, C the chlorinity or salinity, Q the (time-dependent but cross-sectionally averaged) discharge, D a dispersion coefficient and S the summation of source and sinks. The dispersive part of the transport is given by $A \cdot D \cdot \partial C/\partial x$. To include the (time-averaged) variation-transport determined from the 3D-model results in the dispersive transport within the 1D-model, we need to determine a dispersion coefficient. For this we also need the concentration gradient along the estuary. Here we use the timeand crosssection-averaged gradient determined from the 3D-model results. With this, we calculate D as: with F var the variation-transport. 4.3 Test Figure 7 shows the salt flux contributions determined from the spatial decomposition, together with the timeaveraged, maximum and minimum cross-sectional averaged salinity in the 3D-model results, and the dispersion coefficient calculated with equation 4, all for the test case of the trumpet-shaped estuary with variable depth. The results show an along estuary varying dispersion coefficient with maximum values at about 25 km from the sea side and approaching zero at the river

Figure 7. Determination of dispersion coefficients for 1D-models from 3D-model results. Top) flux contributions from salt

flux decomposition; Centre) time-averaged, maximum and minimum cross-sectional averaged salinity in 3D model; Bottom)

disper-sion coefficient computed from the variation transport.

side, consistent with the variation transport. The values are a bit at the high end, but of the same order of magnitude as values from classical literature listed in Fischer et al. (1979).

5 3D-1D COMPARISON

5.1 Aim & method

Next step is to set-up a 1D-model for the trumpet shaped estuary, to feed this with the calculated dispersion coefficient, to run simulations with this model using a forcing similar to the 3D-model simulations, and to compare the 3Dand 1D-model results. Aim of this exercise is to determine whether we can indeed obtain ‘equivalent dispersion’ in the 1D model, as reflection of the behaviour of the 3D-model. Because we used a time-averaged flux as starting point in section 4, we need to evaluate ‘equivalency’ from time-averaged concentration profiles from the 1D-model.

5.2 Test

As first test in this still on-going project, we set-up the model and ran simulations with the numerical modelling system Sobek 3 (more precise: Sobek 3.4.0, see

Deltares, 2015). The results are shown in Figure 8. Looking to the time-averaged salinity along the

estuary, we see that the 1D-model reproduces the 3D-model results rather well: in the two models the salinity intrusion length is nearly identical, the profile along the estuary has the same shape, and the numerical values are close to each other. We see a slight underestimation of salinity at 5–55 km distance from the mouth, and a slight overestimation at 60–90 km. The differences are slightly larger for the maximum and minimum concentration: the 1D-model overestimates the minimum concentration landward of km 40, and underestimates the maximum concentration seaward of km 65. This behaviour could be explained by the use of a time-independent dispersion coefficient in the 1D-model, while actually especially in the computation of minimum and maximum concentration the time-dependency of 3D-mechanisms that need to be accounted for in 1D-models through dispersion could play a role. By the way, investigation of the importance of time-variation in dispersion could be a useful application of the decomposition instrument. All over, based on these positive results, we expect that 1D reproduction of 3D-model results for salinity can be achieved using this salt flux decomposition method also for more complex estuaries. 6 DISCUSSION Decomposition of salt fluxes is in principle not new. Probably the biggest advance compared to literature is in the technical developments of coupling of the

Figure 8. 3D-1D comparison: time-averaged, maximum and minimum cross-sectional averaged salinity along the estuary for

the trumpet shape test case. Continuous lines: 3D-model results (Delft3D-Flow), dashed lines: 1D-model results (Sobek 3).

decomposition to 3D-model results. With this, a spa

tial image of the importance of contributions can be

obtained and 3D-1D equivalence can be developed.

At present, the decomposition takes place as post

processing to 3D-model output. However, we consider

online decomposition (i.e. while a model is running)

within the code of the modelling system Delft3D-flow.

With that, valuable interpretation of model results in terms of processes could be obtained directly without the need for large amounts of additional model output. When decomposing velocity and salinity, it is a question what is the best choice for decomposition to get the most straightforward connection of the result ing salt flux terms to known physical mechanism. We did not go in the details of that within this paper. But obviously, separation of the cross-sectional variation in variation over the depth and over the width is needed to be able to distinguish e.g. effects of tidal straining from tidal trapping, and to use salt flux decomposition in classification of estuaries. This needs to be added in the future. Another issue is the actually assumed linear super position of event and tide influence in the temporal decomposition. This is an assumption of which the actual limits and consequences have to be elaborated. We foresee that the diagnostic abilities of a salt flux decomposition instrument are most useful for sci entific application. However, for application within engineering practice, the possibilities for verification and improvement of 1D-models can be very useful. Derivation of equivalent dispersion can be used to determine the effect of changes in conditions or sys tem characteristics on dispersion coefficients. Thus, the applicability of 1D models can be extended beyond

the situation of there calibration. This is important,

because this type of models is, although ‘old’and with

limited process descriptions, often applied in engineer

ing practice to determine probability of exceedence

etcetera, because it allows for large numbers of sim

ulations. However, it should be noted that the quality

of dispersion derived though salt flux decomposition

form 3D-models will be determined by the quality of

Fischer H.B., List, E.J., Koh, R.C.Y., Imberger, J. & Brooks, N.H. 1979. Mixing in inland and coastal waters. NewYork Academic Press.

Geyer, R.W. & MacCready, P. 2014.The estuarine circulation. Ann. Rev. Fluid Mech. 46: 175–197.

Godin, G. 1972. The Analysis of Tides. Toronto: University of Toronto Press.

Kranenburg, W.M., Van der Kaaij, T., Van den Boogaard, H.F.P., Verploegh, D., Dijkstra, Y.M., Uittenbogaard, R.E. 2015. Ontwikkeling ‘Zoutfluxontrafelaar’ (fase 1). Deltares-report 1220070-000, Delft (in Dutch). Lerczak J.A., Geyer, W.R. & Chant, R.J. 2006. Mechanisms driving the time-dependent salt flux in a partially stratified estuary. Journal of Physical Oceanography. 36: 2296–2311. Walters, R.A. & Heston, C. 1981. Removing Tidal-Period Variations from Time-Series Data Using Low-Pass Digital Filters. Journal of Physical Oceanography. 12: 112–115. WWAP (United Nations World Water Assessment Programme). 2015. The United Nations World Water Development Report 2015:Water for a SustainableWorld. Paris, UNESCO.
Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

Density currents flowing up a slope

L. Ottolenghi & C. Adduce

Department of Engineering, Roma Tre University, Rome, Italy

R. Inghilesi

Institute for Environmental Protection and Research, Italy

V. Armenio

Department of Engineering and Architecture, University of Trieste, Trieste, Italy

F. Roman

Iefluids, Trieste, Italy

ABSTRACT: The dynamics of lock-release density currents propagating up a sloping bottom are investigated

by Large Eddy Simulations (LES). The flow dynamics are deduced by the analysis of the density and velocity

fields obtained by LES.The density current, during its propagation, is observed to develop different flow regimes:

a slumping phase followed by a self-similar phase. A decrease in the velocities of the current is observed with

the increase of the angle between the bottom boundary and the horizontal direction, θ. A smoother behaviour of

the current profile is also visible for high values of θ. The presence of a backward flow close to the bottom of the

domain is detected and it is found to depend on the inclination of the bottom θ. An accumulation of dense fluid in

the lock region of the tank caused by the reverse flow is also observed. Entrainment processes occurring between

the ambient fluid and the dense current are observed and investigated. In particular, during the propagation of

the current, light ambient fluid is entrained by the dense current, which increases in volume. It is found that the

entrainment is affected by the inclination of the bottom and, in particular, a decrease of the entrainment with the

increase of the steepness of the bottom is observed.

1 INTRODUCTION

Gravity currents are flows in which the predominantly horizontal buoyancy gradient causes the propagation of a dense fluid into another of lower density. Gradients in the temperature or in the concentration fields gen erate the difference in density, and this circumstance widely occurs in nature both in the atmosphere and in the oceans. For this reason, gravity currents have been largely studied by both laboratory experiments and numerical simulations (Adduce et al. 2012; Ottolenghi et al. 2016a, 2016b). Instances of this kind of flows are sea breeze fronts, the discharge of fluvial plumes into the sea at the river mouth (Inghilesi et al. 2012) and oceanic overflows, which are thoroughly discussed in Simpson (1997), together with a large number of other examples. The lock-exchange technique is a common practice applied in the laboratory in order to generate gravity currents by the sudden release of a fixed volume of dense fluid into another ambient one of a lower density. This technique is based on the sudden interaction of two fluids in a tank divided by a vertical barrier. When the barrier is quickly removed, a gravity current forms and the dense fluid starts spreading on the bottom of the tank under the ambient fluid, moving away from the lock region. The flow develops and a defined structure of the current is observed, namely a head followed by a body and a tail region (Cantero et al. 2007). KelvinHelmholtz billows and lobe-and-cleft instabilities are also observed: the first related to the velocity shear at the interface between the two fluids, and the latter caused by the intrusion of ambient fluid under the nose of the dense current. During the propagation of the gravity current, different flow regimes are generally identified (Rottman and Simpson 1983). Suddenly after the removal of the vertical barrier, the front of the gravity current propagates at a constant velocity, and this first regime is commonly referred to as slumping phase. It is followed by a self-similar phase, during which the front of the current slows down and evolves according to the theoretical power law of $t^{2/3}$. Finally, a third viscous phase with the front of the current decelerating as $t^{1/5}$ can occur if the viscous forces become significant (Huppert 1982). During the propagation of a gravity current, mixing between the dense and the ambient fluids occurs. In fact, ambient fluid is entrained by the current, mixing with it and affecting the flow dynamics. For this reason the entrainment was studied and parametrized by different authors (Turner 1986; Parker et al. 1987; Cenedese and Adduce 2008). In the studies of Cenedese andAdduce (2010) the entrainment in steady gravity currents was parametrized in dependence of the Froude and Reynolds numbers. On the other side, the entrainment in unsteady gravity currents, which develops from the lock-exchange configuration, is still an open issue deserving more research. Different numerical methods were applied in order to study the dynamics of gravity currents. Direct Numerical Simulations (DNS) and Large Eddy Simulations (LES) were performed in order to study the dynamics of gravity currents by means of high-resolution numerical models solving the three dimensional Boussinesq form of the Navier-Stokes equations (Hartel et al. 2000; Ooi et al. 2009; Tokyay et al. 2011; Ot-tolenghi et al. 2016). Recently, a new modelling approach named lattice Boltzmann method was also introduced for the investigation of gravity currents (LaRocca et al. 2013). The lock-exchange configuration is frequently reproduced numerically in order to gain detailed density and velocity fields of this kind of flows. In the present study the dynamics of lock-release gravity currents propagating along an up-sloping bot tom are studied by LES. The main dynamics of the gravity currents are investigated through the inspec tion of the high-resolution density and velocity fields yielded from the numerical simulations. The entrain ment of ambient fluid during the flow development is also evaluated in order to quantify mixing.

2 NUMERICAL FORMULATION

2.1 Problem formulation

The lock-exchange configuration is numerically repro duced in order to investigate the dynamics of unsteady gravity currents spreading along an up-sloping bottom (Figure 1). The numerical domain is characterized by a length L along the spanwise direction, a height H in the bottom wall-normal direction and a width d in the spanwise one. A vertical barrier is located at a distance x_0 from the left wall of the tank in the streamwise direction,

in order to divide the tank in two volumes. Ambient

fluid at density ρ_0 fills the volume on the right-hand

side of the barrier; dense fluid at density $\rho_1 > \rho_0$ fills

the volume on the left-hand side. Both volumes are

filled up to the same height H , and full-depth gravity

currents are generated. The initial aspect ratio of the

lock volume $R = H/x_0 = 1$ is fixed in all the cases

tested. The inclination of the bottom boundary, θ, is

varied in order to investigate the effect of an up-sloping

bottom on the flow dynamics. Three numerical simulations namedTEST1÷TEST3

are performed varying θ. Figure 1. Sketch of numerical configuration. Table 1. Numerical simulations performed. θ U Re Re b Case [∘] [m/s] TEST1 0 0.091 9082 48522 TEST2 1.4 0.081 8098 48522 TEST3 2.5 0.073 7327 48522 The initial buoyancy gradient driving the motion is kept constant in all the simulations, fixing $\rho_0 = 1000$ kg/m 3 and $\rho_1 = 1030$ kg/m 3 . The initial reduced gravity g'_0 , is defined as and in the present study $g' = 0.29$ m/s 2 . A bulk Reynolds number Re and a buoyancy Reynolds number Re_b can be defined as where U is the bulk velocity of the current (i.e. the ratio between the total distance travelled by the current and the duration of each run). ν is the kinematic viscosity and u_b is the buoyancy velocity, defined as The parameters characterizing the cases tested are reported in Table 1. 2.2 Numerical model The numerical model of Armenio and Sarkar (2002) is here applied in order to investigate the dynamics of gravity currents propagating up a sloping bottom. The model is based on the Boussinesq form of the filtered

Navier-Stokes equations (the overbar in the following

equations denotes the LES filtering operation):

where u_i are the velocity components along the x, y

and z directions of the computational domain, corre sponding to the streamwise, the bottom wall-normal and the spanwise directions, respectively. p and s in (6) and (7) are the hydrodynamic pressure and the salinity, respectively. ν and k_s represent the kinematic viscosity and the molecular salt diffusivity. ρ' is the variation of density with respect to the reference value ρ_0 (corre sponding to the reference salinity s_0). Since the field is isothermal, the state equation reads as:

withβ the salinity contraction coefficient.The inclined bottom is numerically modelled by separating the gravity acceleration g in two components (δ_{ij}), ori ented along the x-axis and the y-axis, respectively.
Following the LES approach, the large scales of the motion are directly solved through the integration of the governing equations, while the small and dissi pative scales of turbulence are modelled as subgrid stresses (SGS) (Pope 2000). A dynamic Smagorinsky eddy viscosity model is

applied in order to solve the SGS momentum and salinity fluxes τ_{ij} and λ_j in (6) and (7), respec tively. For the estimation of the SGS, the constants of the model are calculated through the Lagrangian approach of Meneveau et al. (1996). The governing equations are integrated applying the semi-implicit fractional-step method of Zang et al. (1994). The second-order AdamsBashforth technique solves the time advancement of the convective terms. The diago nal diffusive terms are calculated through the implicit Crank-Nicolson scheme, while the spatial derivatives are discretizes by a second-order centered scheme. A multi-grid SOR-algorithm is applied for the calcula tion of the pressure term. Additional details about the numerical model are given in (Armenio and Piomelli 2000; Armenio and Sarkar 2002). The time step of the simulations is calculated assuring a constant value of the Courant number equal to 0.6. The Schmidt number is fixed at the salt water reference value, i.e $Sc = 600$. The domain has dimensions $L = 3$ m and $H = d = 0.2$ m. The grid spacings are $\Delta x = 0.01 H$, $\Delta z = 0.016 H$ and Δy ranges from $0.01H$ at the top, to $0.002H$ at the bottom of the domain (Tokyay et al. 2011). Flat, no-slip boundary conditions are set in the streamwise direction and at the bottom of the tank. A shear-free boundary condition is applied at the top boundary ($y = H$). Periodicity is employed in the spanwise direction and thus no frictional effects related to the presence of lateral walls are simulated; this condition is chosen in order to simulate gravity currents of large relative width, which are closer to the ones occurring in the environment. Zero flux of the scalar is imposed at all the boundaries. The flow field is initialized with the fluid at rest everywhere. A spatial distribution of the scalar is imposed at $t = 0$, with a discontinuity located at $x = x_0$: $\rho = \rho_1$ for $x < x_0$ and $\rho = \rho_0$ for $x > x_0$. 3 RESULTS 3.1 Flow fields Three different numerical simulations are performed varying the inclination of the bottom boundary: $\theta = 0^\circ$, $\theta = 1.4^\circ$ and $\theta = 2.5^\circ$. The dimensionless density field (ρ^*) is defined as Since the flow can be considered mainly twodimensional, averaged quantities along the spanwise direction of homogeneity (ρ^*) and (u) are considered. In order to consider the dense current as most of the fluid that is not purely ambient fluid, the isopycnal (ρ^*) = 0.02 is chosen to define the interface between the gravity current and the ambient fluid (Nogueira etal. 2013b, 2013a). The evolution in time of the front position of the gravity current is here discussed to analyse the different flow regimes. The front position of a gravity current, x_f , is defined as the location along the streamwise direction of the foremost point of the nose of the current. The velocity of the front of the gravity current U_f is evaluated as the time derivative of $x_f(t)$: Figure 2 shows the evolution in time of the front positions for the cases tested. The gravity currents develop two different flow regimes. The first slumping phase characterized by a constant value of the front velocity can be detected by the constant slope of $x_f(t)$ in Figure 2a, visible up to the distance of about nine lock-lengths ($x_f = 1.8$ m). The shallow water theory, developed for gravity current propagating along horizontal boundaries, predicts that during the following

Figure 2. Time evolution of the front position: (a) x_f versus

t; (b) U_f versus x_f.

self-similar phase the front position slows down fol

lowing the theoretical power law proportional to $t^{2/3}$.

The decrease of the front velocities for all the cases

simulated is clearly visible in both Figures 2a and

2b. The effect of the up-sloping bottom can be seen

through a reduction of the velocities of the current.

In fact, in Figure 2 the curves referring to the sloping

cases are underneath the horizontal case, indicating

that both x_f and U_f are affected by θ. In particular, after

the transition from the slumping to the self-similar

phase, the different $x_f(t)$ diversify and the up-sloping

cases decelerate more abruptly than the horizontal

case. The presence of an up-sloping bottom affects not

only the front propagation of the gravity current, but

the entire behaviour of the flow. In fact, a reverse flow

is observed to develop close to the bottom boundary, in the tail region of the gravity current (in agreement with Lombardi et al. 2015). Part of the dense fluid, due to gravity, detaches from the body of the current and starts to flow backward, forming an accumulation region in correspondence of the lock. Furthermore, the interface between the dense and the ambient fluids becomes smoother with the increase of θ, and the height of the gravity current decreases. The density field of TEST3 is shown in Figure 3 together with the streamwise component of velocity at different times. The first time presented (Figures 3a and 3b) reports the current's behaviour during the slumping phase. High values of $\langle \rho * \rangle$ are visible in the head and in the body regions of the gravity current, while $\langle \rho * \rangle$ decreases in the tail region (Figure 3a). From the inspection of (u) in Figure 3b, it is possible to observe that while the dense current is flowing downstream, the ambient fluid above flows backward for continuity. Furthermore, Kelvin-Helmholtz billows develop at the interface between the two layers and the motion is still unaffected by the upsloping bed. At later times (Figures 3c and 3d), a backward flow is clearly visible in the tail region of the gravity current: $\langle u \rangle < 0$ at $x \sim 0.7$ m (Figure 3d). Moreover, dense fluid starts to accumulate at the beginning of the tank (Figure 3c), increasing the thickness of the tail of the current. Finally, at the end of the simulation (Figures 3e and 3f), a reduction of (u) in the head and body regions is detectable, the head of the dense current is thinner and the dense fluid stratifies in the lock region. 3.2 Entrainment evaluation During the propagation of a gravity current, ambient fluid is entrained by the dense current, which increases its volume. An entrainment discharge can be defined following the variations in volume of the gravity current�V at each time t with respect to the initial volume of the lock fluid V_0 at the initial time t_0 : where �V(t) and �t are the variation in volume of the dense current at each t and the time elapsing from t_0 to t, respectively. The entrainment velocity, $W_e(t)$, is evaluated as the flow crossing the interface S(t) dividing the dense and the ambient fluids: The entrainment parameter E(t) is the entrainment velocity made dimensionless by a velocity scale chosen equal to 2 U (Turner 1973) and it is defined as where U (t) is the bulk velocity of the current at a certain time t. The entrainment parameter is shown in Figure 4 for all the cases tested. The entrainment parameter decreases as the gravity current advances, until it reaches values of the order of 10^{-2} (in agreement with

Figure 3. Density and velocity fields for TEST3 at different times: (a) $\langle \rho^* \rangle$ at t = 10 s; (b) $\langle u \rangle$ at t = 10 s; (c) $\langle \rho^* \rangle$ at t = 20 s;

(d) $\langle u \rangle$ at t = 20 s; (e) $\langle \rho^* \rangle$ at t = 37 s; (f) $\langle u \rangle$ at t = 37 s. Isopycnals in the density fields refer to $\langle \rho^* \rangle$ = 2%, $\langle \rho^* \rangle$ = 10%,

$\langle \rho^* \rangle$ = 20% and $\langle \rho^* \rangle$ = 50%.

Nogueira et al. 2014 and Ottolenghi et al. 2016a). A

decrease of E with the increase of θ is also visible.

In fact, during the entire development of the gravity

currents, E values referring to the upsloping cases are

lower than the ones evaluated for the horizontal case.

In fact, lower values of the front velocity are observed

for high θ, as well as a smoother interface dividing

the two layers, indicating a reduction of turbulence in

the flow field and a reduction of the interfacial mixing

layer between the two fluids. The entrainment parameter is known to be strongly

dependent on the Froude number representative of

the flow (Turner 1986). Further, previous studies

(Cenedese and Adduce 2008, 2010) observed the

dependence of the entrainment parameter on both the

Froude and the Reynolds numbers. Defining g' as the

mean value of the reduced gravity betweenthe values

assumed at the beginning (g'_0) and at the end (g) of each simulation, it is possible to evaluate the Froude number as The final value of the entrainment parameter is shown in Figure 5 versus the product Re Fr. The horizontal case (TEST1), marked by the circle, is located at the top-right side of the figure, indicating high values of FrRe and E. On the contrary, the star marker referring to the case $\theta = 2.5^\circ$ (TEST3), is placed on the low-left side of the figure, indicating reduced values of FrRe and E. In fact, both Fr and Re decreases increasing θ, because the flow velocity is reduced, with the consequent decrease of E. These findings are in agreement with Cenedese and Adduce (2008) and Cenedese and Adduce (2010). Thus, the amount of ambient fluid

Figure 4. Entrainment parameter versus the front position.

Figure 5. E_f versus the product ReFr.

entrained by the dense current during its propagation

is affected by the presence of an upsloping bottom: the

values of E decrease with the increase of θ.

4 CONCLUSIONS

Lock-exchange gravity currents propagating along an

upsloping bottom are investigated by LES.The inclina

tion of the slope θ is varied and its implications on the

flow dynamics and on the entrainment processes are

analysed and discussed. These kind of flows widely

occur in the environment and for this reason are of

relevant interest for environmental and engineering

sciences. For example, oceanic overflows or seismo

genic underflows following an earthquake can prop

agate over complex bathymetries, and their velocity

of propagation and density concentration are essential

Huppert, H. (1982). The propagation of two-dimensional and ax-isymmetric viscous gravity currents over a rigid horizontal surface. J. Fluid Mech. 121, 43–58.

Inghilesi, R., L. Ottolenghi, A. Orasi, C. Pizzi, F. Bignami, & R. Santoleri (2012). Fate of river tiber discharge investigated through numerical simulation and satellite monitoring. Ocean Science Discussions 9(2), 1599.

La Rocca, M., P. Prestininzi, C. Adduce, G. Sciortino, & R. Hinkelmann (2013). Lattice boltzmann simulation of 3d gravity currents around obstacles. International Journal of Offshore and Polar Engineering 23(3), 178–185.

Lombardi,V., C.Adduce, G. Sciortino, & M. L. Rocca (2015). Gravity currents flowing upslope:laboratory experiments and shallow water simulations. Phys. Fluids 27(1), 016602.

Meneveau, C., T. Lund, & W. Cabot (1996). A lagrangian dynamic subgrid-scale model of turbulence. J. Fluid Mech. 319, 353–385.

Nogueira, H., C. Adduce, E. Alves, & M. Franca (2013a). Analysis of lock-exchange gravity currents over smooth and rough beds. J. Hydraul. Res. 51(4), 417–431.

Nogueira, H., C. Adduce, E. Alves, & M. Franca (2013b). Image analysis technique applied to lock-exchange gravity currents. Meas. Sci. Technol. 24(4), 047001.

Nogueira, H., C. Adduce, E. Alves, & M. Franca (2014). Dynamics of the head of gravity currents. Environ. Fluid Mech. 14, 519–540.

Ooi, S., G. Constantinescu, & L. Weber (2009). Numerical simulations of lock-exchange compositional gravity current. J. Fluid Mech. 635, 361–388.

Ottolenghi, L., C. Adduce, R. Inghilesi, V. Armenio, & F. Roman (2016a). Entrainment and mixing in unsteady gravity currents. J. Hydraul. Res., DOI: 10.1080/00221686.2016.1174961. Ottolenghi, L., C. Adduce, R. Inghilesi, V. Armenio, & F. Roman (2016b). Les investigation on entrainment in gravity currents. In River flow 2016 Proceedings, 8th International Conference on Fluvial Hydraulics, St. Louis, MO. Ottolenghi, L., C. Adduce, R. Inghilesi, F. Roman, & V. Armenio (2016). Mixing in lock-release gravity currents propagating up a slope. Phys. Fluids 28(5). Parker, G., M. Garcia,Y. Fukushima, & W.Yu (1987). Experiments on turbidity currents over an erodible bed. J. Hydraul. Res. 25(1), 123–147. Pope, S. B. (2000). Turbulent flows. Cambridge university press. Rottman, J. & J. Simpson (1983). Gravity currents produced by instantaneous releases of a heavy fluid in a rectangular channel. J. Fluid Mech. 135, 95–110. Simpson, J. E. (1997). Gravity currents: In the environment and the laboratory.

Cambridge University Press. Tokyay, T., G. Constantinescu, & E. Meiburg (2011). Lockexchange gravity currents with a high volume of release propagating over a periodic array of obstacles. J. Fluid Mech. 672, 570–605. Turner, J. (1973). Buoyant convection from isolate sources. Buoyancy Effects In Fluids. Cambridge University Press. Turner, J. S. (1986). Turbulent entrainment: the development of the entrainment assumption and its application to geophysical flows. J. Fluid Mech. 170, 431–471. Zang,Y., R. Street, & J. Koseff (1994). A non-staggered grid, fractional step method for time-dependent incompressible navier-stokes equations in curvilinear coordinates. J. Comput. Phys. 114(1), 18–33. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Turbulent entrainment in a gravity current

M. Holzner

Institute of Environmental Engineering, ETH Zurich, Switzerland

M. van Reeuwijk

Department of Civil and Environmental Engineering, Imperial College London, UK

H. Jonker

Department of Geoscience and Remote Sensing, Delft University, The Netherlands

ABSTRACT: We revisit the classical entrainment experiments for gravity currents on inclined slopes (Ellison

and Turner, J. Fluid Mech. 6, 423–448, 1959). We derive an entrainment relation that couples the entrainment

rate E to the production of turbulence kinetic energy, the net effect of buoyancy and inner layer. Using direct

numerical simulations that are run for durations long enough for the flow to reach universal self-similarity,

we show that the net effect of inner layer processes on entrainment is very small and that buoyancy has an

almost negligible effect on E. It is demonstrated that the dominant process causing entrainment is turbulence

production due to shear. Second, we observe that for all simulations the eddy diffusivity and dissipation rate can be parameterised using the turbulence kinetic energy and shear parameter. This information can be used to derive an entrainment law which is in good agreement with the Direct Numerical Simulation (DNS) results. We discuss the potential reasons for why this result is significantly different from experiments and the classical entrainment law introduced by Ellison and Turner.

1 INTRODUCTION

Density currents are important in a variety of settings, ranging from cold water overflows in the ocean to thunderstorm outflows or sea-breeze fronts in the atmosphere and are of importance to many hydraulic engineering applications (e.g. sediment laden gravity currents). A central aspect that controls the dynamics of density currents is the entrainment of external fluid into the turbulent flow. The entrainment in an inclined dense gravity current was first studied by (Ellison & Turner 1959), whose interest was triggered by a very practical problem occurring in coal mines. Numerous other investigations have dealt with the problem since then, and we refer to (Wells et al. 2010) and (Odier et al. 2014) for overviews on the topic. The main parameter that quantifies the entrainment is the “entrainment rate” E, which is the ratio between the entrainment velocity normal to the current w and the mean down stream velocity U , $E = w/U$. Many authors have tried

to determine E as a function of the Richardson number

(which is the ratio between the stabilizing buoyancy

force and destabilizing shear force) either empirically

or based on theoretical modeling (e.g. (Ellison and

Turner 1959, Cenedese and Adduce 2008, Cenedese

and Adduce 2010, Nogueira et al. 2014, Odier et al.

2014, Krug et al. 2015); see also (Wells et al. 2010)

over the years, our understanding of entrainment is still incomplete, hampering progress in the theory and parametrization of gravity currents. In this work we use DNSs of temporal gravity currents and we simulate in large domains and for long times to make sure that self similarity is reached. We demonstrate that the dominant process causing entrainment is turbulence production due to shear. We observe that for all simulations the eddy diffusivity and dissipation rate can be parameterised using the turbulence kinetic energy and shear parameter, which is then used to derive an entrainment law which is in good agreement with the DNS results. 2 THEORY We consider a layer comprising of negatively buoyant fluid on a slope of angle α, as sketched in Figure 1. At time $t = 0$, the layer has depth h_0 and a uniform buoyancy b, where $b \equiv g(\rho - \rho_0)/\rho_0$, g is the gravitational acceleration and ρ_0 is a reference density. As $b_0 < 0$, the fluid will accelerate in the positive x-direction, become turbulent and flow down the slope as a gravity current. Because of the problem setup, the flow will remain homogeneous over the x and y direction, and its statistics will thus only depend on the vertical coordinate z and time t. If the angle $\alpha = 90^\circ$, this case represents a plane wall plume. This situation is closely related to that considered by (Ellison &

Figure 1. Definition sketch.

Turner 1959), who also consider a problem like this

where α can be varied, albeit with the difference that

they consider a spatially developing gravity current.At the wall, a no-flux (Neumann) boundary condition is enforced for buoyancy. For velocity, both no-slip and free-slip conditions will be considered. The incom pressible Navier-Stokes equations in the Boussinesq approximation for the coordinate system shown in figure 1 are given by

Here, x= (x, y, z) with x, y the lateral and z the ver tical coordinate, u= (u, v, w) is the fluid velocity, p the pressure, e g = (− sin α, 0, cos α), ν and κ are the kinematic viscosity and diffusivity, respectively, and b is the buoyancy. It is convenient to work with buoyancy because it naturally encompasses situations where density variations are caused by temperature (atmosphere), by salinity concentrations or by a com bination of both (ocean). We average (1,2) over the homogeneous directions x and y which results in

Integrating (4), (5) over z, we obtain where τ w ≡ ν∂u/∂z| w is the kinematic wall shear stress, and the volume flux q, momentum flux m and total buoyancy B are defined as From (7) it follows that B =−B 0 where B 0 =−b 0 h 0 sin α. Consistent with (Ellison and Turner 1959, Krug et al. 2013) we define the following integral or 'tophat' scales implying that Q = hu T , M = hu 2 T and B = hb T sin α. 2.1 The entrainment relation The entrainment law (a turbulence closure relating the entrainment rate to the velocity u T in the layer) is defined as which, using the definition of h, can be rewritten as (Van Reeuwijk & Craske 2015) This provides important information as it reveals that an explicit equation for the entrainment law can be obtained by combining the integral momentum and mean energy equations. Using the definition for M (8),

it follows that the equation for the total mean kinetic energy is given by

Substitution of (2), (6) into (11) results in

where

Furthermore, the Richardson number Ri and friction

factor c_f are defined as The entrainment law (13) is thus determined by

three processes. First and foremost, the turbulent pro

duction contribution E_{prod} is the prime contributor to

turbulent entrainment. Second, E_{buoy} represents the

influence of buoyancy on turbulent entrainment, and it

is expected that the higher Ri, the larger the reduction

in entrainment. The third term is E_{inner} , which repre

sents the processes in the inner layer, i.e. the processes

that are not associated with turbulent entrainment but

that nevertheless enter the entrainment law because

the processes in the inner layer require energy. Wall

friction, as represented by the friction factor c_f is the

primary term reponsible for a reduction in E. The

mean dissipation rate integral is absorbed in E_{inner}

because in the outer layer the integral can be approxi

mated by u_T^2/h which implies that the term is of order

Re^{-1} where the Reynolds number $Re = u_T h/\nu$. Thus,

at high Reynolds number the outer layer contribution

is expected to vanish and the integral is only set by

inner layer processes.

3 SIMULATION DETAILS

The DNS code solves Eqs (1-2) on a cuboidal domain

and is fully parallelized making use of domain decom

position in two directions. The spatial differential Table 1. Simulation data. FS: free-slip; NS: no-slip. Simulation α BC S2 2 FS S5 5 FS S10 10 FS S10N 10 NS S25 25 FS S45 45 FS S90 45 FS operators are discretized using a fourth order symmetry-preserving central differences, and timeintegration is carried out with an adaptive third order Adams-Bashforth method. Periodic boundary conditions are applied for the lateral directions. At the bottom wall, no-slip conditions are applied for the velocity and a Neumann (no-flux) boundary conditions for buoyancy.At the top. free-slip boundary conditions are applied for velocity and Neumann (no-flux) boundary conditions for buoyancy. The initial conditions are given by u= 1 for z < 1, and b(x, y, z) = b 0 for z < 2. The initial conditions are perturbed using smallamplitude random noise to facilitate the transition to turbulence.The fact that B is conserved implies that B 0 is the main forcing parameter, which is also expressed by the fact that in absence of wall-shear, (2) can be integrated to Q ~ B 0 t. Thus we define a Reynolds number which we keep constant for all the simulations considered here. A reference time scale can be defined as t * = (h 2 0 /B 0) 1/2 4 RESULTS 4.1 No-slip vs free-slip at 10 degrees angle In Fig. 2 (top) we show the time evolution of the entrainment rate for the α= 10 o simulation with noslip (red) and free-slip (blue) at the bottom wall. We see a transition from “forced” at early times to “free” for both cases and it appears that this system forgets its initial conditions fast due to the work of buoyancy. We also performed simulations with different initial conditions and these showed all the same long time behavior. It is thus noted that after an initial transient the flow reaches an equilibrium with constant entrainment rate which is not greatly affected by the bottom wall boundary condition. The bottom panel of Fig. 2 shows the decomposition of the entrainment rate (Eq. 13) for the no-slip (left) and the slip (right) case. We observe that the main contribution to E is due to the production term. This contribution of the production term is larger for the no-slip condition, but its effect is decreased through a negative friction term such that the overall E is similar for both cases. Next we show that in this flow the gradient Richardson number (Fig. 3a), the flux Richardson number (Fig. 3) and the mixing

Figure 2. Entrainment rate and decomposition. a) lines: E

according to direct definition Eq. 10. Symbols: E as determined

from the decomposition (Eq. 13). (b) Decomposed entrainment for no-slip case. (c) Decomposed entrainment for free-slip

case.

Figure 3. (a) Gradient Richardson number $Ri_g = (\partial b/\partial z) \cos \alpha/(\partial u/\partial z)^2$; b) Flux-Richardson number $Ri_f = P_b/P_s$; c) Mixing

efficiency $=$P_b/\varepsilon$. Red is no-slip, blue is free slip. Lines represent different time instances.

efficiency (Fig. 3c) are all remarkably constant in the

mixing layer. Again, there is little effect of the wall

boundary condition on this result.

4.2 Free-slip simulations for a range of angles

In Fig. 4 (top) we show the time evolution of layer

depth (left) and Ri (right) for all DNSs with free-slip

condition demonstrating that self similarity is reached

for all cases. This is further exemplified by the time

collapse of several quantities when rescaled by their

appropriate self-similarity variables, namely stream

wise velocity (Fig. 4 centre, left), buoyancy (Fig. 4 centre, right), as well as shear Reynolds stress ((Fig. 4 bottom, left) and buoyancy flux (Fig. 4 bottom, right). All simulations converge to an approximately constant entrainment rate after rather long transients, see Fig. 5. The smaller the angle the stronger the stratification and therefore the flow becomes more stable and E becomes small. A central result of this work is the behavior of E against Ri plotted in Fig. 4.2. Our data shows an approximately linear decrease to zero for $Ri < 0.25$. The trend is very different from the one of (Ellison & Turner 1959), in particular the observed range in Ri is much smaller.

Figure 4. Self-similarity behavior of various quantities for different simulation cases.

Figure 5. Entrainment coefficient as a function of time showing long transients.

Figure 6. Entrainment rate as a function of Ri, showing that the observed range of Ri is much smaller than the Ellison-Turner

relation. Note that as α→ 0, Ri < 1/4 which is a stability requirement for stratified shear flows.

Figure 7. Quantification of lengthscales. a) '' eε = e 3/2 /ε, '' S = e 1/2 /S, '' N = e 1/2 /N , clearly showing that the shear scale

dominates. b) '' m;ν =−w ′ u ′ /(∂u/∂z)e 1/2 , '' m;κ =−w ′ b ′ /(∂b/∂z)e 1/2 , showing the excellent correlation with '' S . This shows that

the shear-scale is indeed dominating the problem.

4.3 Turbulence parameterisation

In Fig. 4.2 (left) we show turbulence, shear and buoy

ancy length scales demonstrating that the shear length

scale dominates (e.g. (Jackson et al. 2008)). Moreover, we find proportionality between mixing length scales and the shear length scale (Fig. 4.2 right). Turbulent fluxes are thus well approximated by a mixing length model.

5 DISCUSSION

We can use the latter result to devise a mixing length

model and calibrate its free constants from our DNS

to obtain

This entrainment law is in reasonably good agree

ment with the DNS (dashed line in Fig. 4.2). Many

experimental and model investigations address the

spatially developing gravity current. In order to com

pare with the spatially developing gravity current, at least qualitatively, we set

and thus

From the simulations the universal entrainment regime

is observed beyond $t/t* = 20$. If the results are directly

transferable this indicates that most experimental mea

surements have been conducted too close (generally

taken at about $x/h_0 = 10$) to the source to pick up the

“free” or “equilibrium” entrainment behaviour which

could explain why (Krug et al. 2015) and (Odier et al.

2014) measure much higher values of E than observed

Remote sensing and coastal morphodynamic modelling: A review of current

approaches and future perspectives

Mario Benincasa

Department of Engineering, Università Roma Tre, Roma, Italy

Federico Falcini

Institute of Atmospheric Sciences and Climate, CNR, Italy

Claudia Adduce

Department of Engineering, Università Roma Tre, Roma, Italy

Rosalia Santoleri

Institute of Atmospheric Sciences and Climate, CNR, Italy

ABSTRACT: We want to investigate the use of remotely sensed data in shoreline morphodynamic modelling.

For this aim, we present a review of current approaches for Total Suspended Matter (TSM) retrieval from

satellite data in coastal waters and outline some perspectives for the definition of an operative, satellite-based,

alongshore sediment flux. Coastal, turbid and optically complex waters have traditionally posed challenges to remote sensing and space-borne techniques for a variety of reasons: the complexity and variability of their radiative processes on one hand; spatial, temporal and spectral resolution issues on the other. Nevertheless, different sensors/satellites have been used in order to retrieve the main biogeochemical characteristics of these waters. Geostationary satellites (such as SEVIRI for European seas and COMS1 for South Korea) have been recently adopted for their high temporal resolution while classical ocean colour mappers (such as SeaWiFS, MODIS, MERIS, VIIRS) and other Earth observation satellites (like the Landsat series, Envisat, ERS series and Sentinel program) are commonly used because of their finer spatial and spectral resolutions. The immediate future looks promising because of the recent launch of the first satellite of the Sentinel-3 constellation with a new ocean colour instrument (OLCI) (21 spectral bands, 300m spatial resolution and a revisit time of 1.4 days). In the longer term, from 2018 onwards, the launch of Hyper Spectral instruments (e.g., German EnMAP, Italian PRISMA) with much higher spectral resolution capabilities, although at the expenses of a lower time resolution, promises finer water constituents determination and grain size characteristics, based on their spectral signatures. Consequently, a large variety of different methods, empirical or semi-analytical, are used by the scientific community in order to get the most suitable algorithm for the TSM retrieval. Envisaged developments include multi sensor approaches, new algorithms, coupled sea-atmosphere radiative transfer models. We here discuss the best strategy in order to achieve the most suitable regional TSM product for coastal geomorphologic applications and thus to pair it with coastal water velocity fields, allowing for a satellite-based definition of alongshore sediment transport. We also discuss spatial, temporal, and spectral characteristics of different sensors and novel algorithms that might combine these properties together.

1 INTRODUCTION

In the scope of coastal morphodynamic modeling, we want to investigate the use of remotely sensed data to evaluate the physical quantities needed by sediment transport models in coastal environments. Shoreline morphodynamics is generally modelled by means of the Exner equation, which describes the conservation of mass that relates the seabed height (η) and the sed iment suspended in and transported by water (Paola and Voller 2005): where □q_s is the sediment flux. If we are interested only in the shoreline profile, we can rewrite Exner equation in one dimension: where η now represents the position of the shoreline as in Figure 1 (Ashton et al. 2001, Ashton and Murray 2006). The evaluation of the sediment flux ($Q_s(x, t)$) is usually a challenging task, subject to heavy assumptions and approximations (Komar 1971, Rosati et al. 2002), where, in general, only a “prevalent” wave field and a “typical” grain size are considered.

Figure 1. Alongshore sediment transport. Modified after (Ashton et al. 2001).

We here explore the possibility to evaluate the sedi ment flux from a remote sensing approach. We aim at combining a satellite-inferred sediment concentration together with a water velocity field, as obtained by e.g. coastal oceanographic models. With such a combina tion we can obtain a sediment flux and we can feed this sediment flux in a coastal morphodynamic model.

2 THE REMOTE SENSING PROBLEM

Ocean remote sensing (Mobley et al. 2011, Robinson 2004) refers to the use of optical measurements, typi cally made from space satellites, to obtain information about the water body under observation: constituents, IOPs (Inherent Optical Properties), geometry. Sig nals from the UV region (wavelengths in the range 300 ÷ 400 nm) to the microwave region are used, both in passive and active sensors. The Ocean Color Radiometry, commonly called "Ocean Color Remote Sensing", typically deals with visible (wavelengths in the range 400 ÷ 700 nm) and near-IR (wavelengths in the range 700 ÷ 2000 nm) light, captured by pas sive sensors. Since 1978, when the launch of CZCS (Coastal Zone Color Scanner) started the passive ocean color era, different sensors with different fea tures have been put on different orbits around the globe. Figure 2, from (Mobley et al. 2011), shows the spectral bands used by some ocean color sen sors. Different sensors show different capabilities in terms of spatial, temporal, spectral and radiometric resolutions and radiometric dynamic range.The differ

ences are due to technology, orbit (e.g. geostationary

vs polar), design considerations and intended use (e.g.

operational vs scientific). Passive ocean color remote sensing is conceptu

ally simple: sunlight, whose spectral and geometric

properties are known, enters Earth's atmosphere, tra

verses it, enters the water body, leaves it, traverses the

atmosphere again and finally reaches the sensor. In all

these media the light field is absorbed, reflected, and
Figure 2. Wavelength bands used by various ocean color remote sensors. From Mobley et al. 2011. Figure 3. Contributions to the total upwelling radiance L_u . L_a (orange arrows) is the atmospheric path radiance; L_r (red arrows) is the surface reflected radiance; L_w is the water leaving radiance; L_t is the transmitted radiance, in the water. Yellow arrows represent the sun's unscattered beams. Thick arrows represent single scattering, thin ones multiple scattering contributions. From Mobley et al. 2011. scattered in a wavelength dependent way. Every possible constituent of both the atmosphere and the ocean shows its own spectral behavior in response to incident light. Figure 3, from (Mobley et al. 2011), outlines the different processes that contribute to the light field that reaches the sensor. The "forward problem", i.e. the determination of the light field, given the light sources, the inherent optical properties (IOPs) of the water and the atmosphere, and the physical boundaries, can be regarded as a solved problem.The radiative transfer theory gives us the Radiative Transfer Equation, which can be derived in different ways. In its more general form, the RTE can be written as:

where$\sigma_{ext} = \sigma_{abs} + \sigma_{scat}$ is the extinction cross section,

due to both absorption and scattering. The equation 3

simply states that the change in field intensity I while

propagating for a ds length in a given direction is the sum of the losses due to absorption and scattering out of the beam and the gains from scattering in and light sources. The source term j for elastic scattering in the beam assumes the form:

where ▯^ is the direction (versor) of the incident beam and ▯^ ′ the direction of scattering, and P(▯^, ▯^ ′) is the phase function. The solution of the RTE can be done analytically, for very simple cases, or numerically, by means of different tools, e.g. (Emde et al. 2015). The remote sensing problem is, on the other hand, an inverse problem: given radiometric measurements of water-leaving light fields, determine the inherent optical properties of the water body, and from these the water’s constituents. As such, an inverse problem poses issues on both the uniqueness of the solution and on its stability, i.e. its sensitivity to errors in radiomet ric measurements. Although it can be shown that if we have a perfect measure of the light field I (x, y, z), then there is a unique solution to the inverse RTE, this is of course useless in practical applications: if we could measure the full light field then we could also directly measure the IOPs we want to discover. In practice, we have only a limited set of imperfect light field measurements from which we want to retrieve as much information as possible. Therefore, we have to cope with severe limitations and large errors in the results we obtain. Over the years, several techniques have been developed to cope with the inherent dif ficulties of such a problem. No technique is perfect, many of them are experimental and limited to partic ular basins and conditions, but each has demonstrated its own value in its field of application. The classical approach of the ocean color science is

to apply an atmospheric correction to the total radiance as received by the sensor, L_u in figure 3, in order to remove the contributions of the atmospheric path radi ance L_a and of the surface reflected radiance L_r, and isolate the water leaving radiance L_w. The water leav ing radiance is then normalized (solar zenith angle, ozone layer, . . .) and converted to a reflectance value called the “remote sensing reflectance”:

where E_d is the down-welling solar irradiance at the sea surface. All these quantities are usually intended as spectral quantities. The solution of an inverse problem often requires

assumptions and some a priori knowledge of the solu tion itself. (Morel & Prieur 1977) proposed a first broad division of the water bodies under investigation based on their spectral behavior, that evolved to the following definitions (Mobley et al. 2004): • case 1 waters, open ocean waters, are those dominated by phytoplankton and related colored dissolved organic matter (CDOM) and detritus degradation products • case 2 waters are everything else, namely waters whose optical properties are significantly influenced by other constituents such as mineral particles, CDOM, or microbubbles, whose concentration do not covary with the phytoplankton concentration. See (Mobley et al. 2004) for a discussion on the validity and applicability of such classification. The various techniques and the algorithms that have been developed, basically ingest the previously defined spectral remote sensing reflectance values ($R_{rs}(\lambda)$) at different λ-s and produce an esteem of a geophysical, biological, or chemical variable of interest (e.g. chlorophyll concentration, SST, . . .). Again we have a first broad division of these algorithms: “band arithmetic” algorithms and “inversion” or “analytical” algorithms, see (Odermatt et al. 2012) for a review of different algorithms focused on case 2, optically complex, waters. The first category algorithms are statistical regression methods that try to relate the variable under investigation (e.g. the chlorophyll concentration or the TSM) to the reflectances at different wavelengths. Linear, nonlinear, polynomial regression techniques are used, as well as machine learning techniques (artificial neural networks, support vectors machines). This kind of algorithms are easy, light on computing resources, but are usually limited to a region and/or a season. The second ones, on the other hand, try to solve the inverse RTE problem, by solving forward (possibly simplified) RTE problems. Even if, sometimes, parameters are chosen by means of some sort of regression, this kind of algorithms is supposed to “capture” the physics of the light-matter interaction and it is expected to be less regional and less seasonal, although at the expenses of a heavier computational load. Actually, their computational load is often their key limiting factor in time-constrained environments, like weather forecast or operational products. Another approach is to consider a unique light field, i.e. not introducing the atmospheric correction concept nor the water leaving radiance. The idea is to address a unique (inverse) RTE problem with a medium change at the sea surface. This appears especially interesting for our problem, as we will see. For our application, the water constituent we are interested in is the Total Suspended Matter (TSM), i.e. the concentration, measured in $g \cdot m^{-3}$ (or, equivalently, in $mg \cdot L^{-1}$), of suspended material in the water. TheTSM is characterized by both its concentration and its granulometry. Typical values for TSM concentration in coastal waters lie in the range $10 \div 100$ gm^{-3}, (D’Sa et al. 2007, Myint and Walker 2002). On the other side, the particles’ size distribution conditions the actual morphodynamic effect of the suspended sediment. For their morphdynamic effect we should consider only the particles in the 2 ÷ 250 μm size range, i.e. the silt and fine sand classes of particle sizes, (Rosati et al. 2002, Komar 1971). The particles'

Figure 4. Different scattering approximations for different size parameters $x = 2\pi r\ \lambda$; graph of particle radius (and type on the right-hand axis) and radiation wavelength partitioned by different size parameters. Credit: W. Brune (after Grant Petty).

size distribution affects also the interaction between light and matter, as roughly described by the size parameter x that relates the radius r of a particle and the wavelength of the incident light. Different ranges of size parameter allow for different scattering approximations that can be used when solving the RTE, as graphically represented in Figure 4. (Kostadinov, Siegel, & Maritorena 2009) shows an algorithm for the retrieval of particle size distribution from multispectral (SeaWIFS) satellite observations. When the size of particles suspended in water is comparable to the size of scattering material in atmosphere, it could be interesting to treat the radiative transfer problem as a single entity and not as separate transfers in water and in atmosphere. Calibration of an algorithm with regional datasets always poses the question of its generality or its regionality. TSM features (particle size distribution, concentration, refractive index i.e. composition) may vary greatly from region to region and even among dif ferent seasons in the same region. The non-uniqueness of the inverse RTE solution thus implies that assump tions and/or calibrations have to be made. Generally speaking, algorithms that are calibrated for a cer tain region give poor results if applied unmodified to other regions, where the TSM features could be quite different. This is of course especially true for regres sion based algorithms, i.e. for algorithm that don't even try to model the light-matter interaction in any way, but that simply try to map observed reflectances to geophysical variables by means of some regres

sion. Nevertheless, attempts have been made to derive Figure 5. Tyr2010 cruise in the Mediterranean sea. Figure 6. Nechad (et al., 2010) TSM algorithms vs ISAC-CNR Tyr2010 cruise data. general TSM algorithm, that could work for different sensors, different regions and different seasons: see (Nechad et al. 2010) for an example. This algorithm has been applied also to the Mediterranean sea, matching it with the TSM measurements performed during Tyr2010 oceanographic cruise by CNR-ISAC (see Figure 5 and Figure 6). (Neukermans et al. 2009) focuses on algorithms for the geostationary data, localized on the North Sea. The data are provided by the SEVIRI sensor on board of the Meteosat Second Generation (MSG) meteorological satellites by Eumetsat. (Neukermans et al. 2012) follows on the previous results to investigate the High Resolutions Visual band (the Panchromatic Sensor) on board of the same satellites to increase the spatial resolution. The use of geostationary satellites is interesting for their inherent high temporal resolution capabilities: basically a picture of the entire half-Earth in their field of view every 15'(SEVIRI) or hour (GOCI).The interest in this high temporal resolution is the possibility to follow rapidly evolving phenomena like river flood events or tides.

3 MAPPING TSM FROM SPACE: SPACE-BORNE SENSORS

In this section we want to present a brief list of the main ocean color sensors that have been or can be used to retrieve TSM data. See Arnone et al. (2005) for an historic report. Second generation Ocean Color sensors, NASA's SeaWIFS (activity period: 1997–2010) and ESA's MERIS (activity period: 2002–2012) have been widely used for the investigation of coastal waters. They have provided two huge datasets for the years they have been active. In particular, within the MERIS project, ESA created the CoastColour project (ESA, Ruddick et al. 2010), devoted to the study of the Case 2 waters exploiting the 300 m spatial resolution time series acquired by MERIS. Building on the MERIS expe rience, the project will continue with the upcoming OLCI sensor.

3.1 MODIS

The MODIS (Moderate Resolution Imaging Spectro radiometer) sensor is on board of the Terra and Aqua spacecrafts, launched December 1999 and May 2002, respectively, flying in sun-synchronous, near-polar, circular orbit. It is a multispectral (36 bands) mapper with a full earth coverage every one to two days. Spa tial resolution ranges from 250 m to 1 km, depending on band. Refer to (NASA 2003) for full sensor speci fications. See (Miller & McKee 2004) for a case study of coastal waters TSM concentration retrieval by using MODIS data 250m resolution (band 1). Good relation ship is reported between TSM concentration and Band 1 (620 ÷ 670 nm) reflectance. The 250 m spatial reso lution is compared to the 1 km resolution available in other bands.

3.2 VIIRS

The VIIRS (Visible Infrared Imaging Radiometer Suite) sensor is on board of the Suomi-NPP satellite, launched October 2011 by NASA, flying on a near polar orbit. VIIRS extends and improves upon a series of measurements initiated by the Advanced Very High Resolution Radiometer (AVHRR) and the Moder ate Resolution Imaging Spectroradiometer (MODIS). Like MODIS, VIIRS is a multi-disciplinary sensor providing data for the ocean, land, aerosol, and cloud research and operational users. VIIRS spectral cov erage will allow for data products similar to those from SeaWiFS as well as SST, a standard MODIS product. SST is an Essential Climate Variable (ECV) and, through validation with instruments traceable to NIST standards, is a Climate Data Record. Also, as with SeaWiFS and MODIS, the VIIRS scan and orbit geometries will provide global coverage every two days. Refere toVIIRS pages on NASA website (NASA

2016, NASA 2015) for further information on the

mission. 3.3 SEVIRI The SEVIRI (Spinning Enhanced Visible and Infrared Imager) sensor is on board of the first Meteosat Second Generation (MSG) satellite, launched February 2003, flying in a geostationary orbit. The geostationary orbit allows for very high temporal resolution (15 minutes), although at the expenses of a lower spatial and spectral resolutions. It can be interesting for the investigation of fast evolving phenomena, like river floods and tides. Refer to (Aminou 2002) for further details on the instrument itself. (Neukermans et al. 2009, Neukermans et al. 2012) show the application of the SEVIRI instruments for the retrieval of regional TSM maps and the study of diurnal turbidity variability in the southern North Sea. 3.4 OLCI The Ocean and Land Color Instrument (OLCI) sensor will be on board of the Sentinel-3 constellation of satellites, Sentinel-3A, Sentinel-3B and Sentinel-3C; the first one has just been launched, the other two are going to be launched by the end of 2017 and before 2020, respectively. Refer to (Malenovsky` et al. 2012) for an introduction to the Sentinel program, and to ESA Sentinel online web site (ESA 2016) for detailed documentation. 3.5 Hyperspectral sensors Sensors with 200+ narrow spectral bands, covering a contiguous or almost contiguous portion of the spectrum, are called hyperspectral. Placed on a near-polar orbit, their spectral resolution promises the ability to detect spectral signatures. In particular, regarding the TSM, one can expect to be able to infer e.g. particle size distribution, or chemical species of the suspended particulate. The high spectral and spatial resolutions of such instruments, usually imply a low temporal resolution. Furthermore these kind of satellites are not mappers designed for operational purposes, but are more research oriented mission. Typically, observation areas have to be “purchased” from the operator. (Staenz and Held 2012) presents a brief report of the past and future hyperspectral earth observation missions. Among the various missions, the most advanced appear to be the PRISMA, by ASI, (Lopinto and Ananasso 2013) and the EnMAP, by DLR. Both should be launched in the 2017–2018 timeframe. 4 TOWARDS THE DEFINITION OF THE SEDIMENT FLUX As seen, different sensors, different approaches and different techniques can be used, and have been used, to infer TSM maps from satellite remotely sensed data. On this respect, we will investigate the use of either polar orbiting (both current and future) ocean colour sensors and geostationary sensors, as well as other Earth Observation sensors, in order to infer TSM

maps for the Mediterranean coastal waters. Current

and new algorithms will be investigated, together with

the possibility to merge the data from different sources,

in order to leverage both the spatial/spectral resolu

tions of polar orbiters and the temporal resolution of

geostationary sensors. Total Suspended Matter (TSM) accounts for both

organic (CDOM, Coloured Dissolved Organic Mat

ter or Gelbstoff or yellow substance) and inorganic

(sediment, mineral) components, which show differ

ent spectral behaviours, i.e. different absorption and

back-scattering features, especially at lower wave

lengths. (Myint and Walker 2002, Doxaran et al. 2005,

D'Sa et al. 2007) investigate band ratio algorithms

for sediment-dominated coastal waters. (Morel and

Bélanger 2006) investigate the possibility to distin

guish the water components by means of multispectral

imagery from SeaWIFS, MERIS, and MODIS sensors.

Spectral resolution plays therefore an important role

in such a task, and we may expect improvements when

hyperspectral sensors will be deployed. Only the sediment, i.e. the mineral part of the

suspended matter, will play a role in coastal morpho

dynamic, and, in particular, only the grains that are in the 2 ÷ 250 μm size range, i.e. from silt to fine sands

(Komar 1971, Rosati et al. 2002). We have now defined a multifaceted problem,

ideally we would like to infer:

• the inorganic part of the TSM

• the mineral specie(s) that make it up

• the particle size distribution

• the vertical distribution (Woz´niak and Stramski 2004) presents a study to

model the optical behaviour of the mineral particles

suspended in seawater, based on Mie scattering the

ory. Their interest here, however, is in filtering out the

sediment contribution to the ocean color, in order to

improve the estimation of the organic component. (Babin et al. 2003, Lubac and Loisel 2007, Martinez

et al. 2015, Park and Latrubesse 2014, Wang and Lu

2010) all address the problem of the discrimination of

the sediment part of the suspended matter in coastal

waters by means of its different optical behaviour in

terms of absorption and scattering. The first two focus

on different European regions, while (Martinez et al.

2015, Park and Latrubesse 2014) focus on theAmazon

river basin and the (Wang and Lu 2010) on theYangtze

River basin. (Kostadinov et al. 2009) proposes a method to

derive Particle Size Distribution parameters from

ocean color observation. Particle size is assumed to be

distributed as per the Junge power law (Junge 1963):

and a method is presented to derive PSD parameters

N 0 and ξ based on Mie theory. In principle, sediment maps are 2-dimensions

Emde, C., R. Buras-Schnell, A. Kylling, B. Mayer, J. Gasteiger, U. Hamann, J. Kylling, B. Richter, C. Pause, T. Dowling, et al. (2015). The libradtran software package for radiative transfer calculations (version 2.0).

ESA. Meris instrument for remote sensing of the coastal zone. http://www.coastcolour.info.

ESA (2016). ESA Sentinel Online. https://earth.esa.int/web/sentinel.

Galappatti, G. & C. Vreugdenhil (1985). A depth-integrated model for suspended sediment transport. Journal of Hydraulic Research 23(4), 359–377.

Junge, C. E. (1963). Air chemistry and radioactivity. 1963, 382.

Komar, P. D. (1971). The mechanics of sand transport on beaches. Journal of Geophysical Research 76(3), 713–721.

Kostadinov, T., D. Siegel, & S. Maritorena (2009). Retrieval of the particle size distribution from satellite ocean color observations. Journal of Geophysical Research: Oceans 114(C9).

Lopinto, E. & C. Ananasso (2013). The prisma hyperspectral mission. In proceedings of the 33rd EARSeL Symposium Towards Horizon 2020.

Lubac, B. & H. Loisel (2007). Variability and classification of remote sensing reflectance spectra in the eastern english channel and southern north sea. Remote Sensing of Environment 110(1), 45–58.

Malenovsky`, Z., H. Rott, J. Cihlar, M. E. Schaepman, G. GarcíaSantos, R. Fernandes, & M. Berger (2012). Sentinels for science: Potential of sentinel-1,-2, and-3 missions for scientific observations of ocean, cryosphere, and land. Remote Sensing of Environment 120, 91–101.

Martinez, J.-M., R. Espinoza-Villar, E. Armijos, & L. Silva Moreira (2015). The optical properties of river and floodplain waters in the amazon river basin: Implications for satellitebased measurements of suspended particulate matter. Journal of Geophysical Research: Earth Surface 120(7), 1274–1287.

Miller, R. L. & B.A. McKee (2004). Using modis terra 250 m imagery to map concentrations of total suspended matter in coastal waters. Remote sensing of Environment 93(1), 259–266.

Miller, R. L., B. A. Mckee, & E. J. D'sa (2007). Monitoring bottom sediment resuspension and suspended sediments in shallow coastal waters. In Remote sensing of coastal aquatic environments, pp. 259–276. Springer.

Mobley, C., E. Boss, & C. Roesler (2011). Ocean optics web book. http://www.oceanopticsbook.info.

Mobley, C., D. Stramski,W. Bissett, & E. Boss (2004). Optical modeling of ocean waters: Is the case 1-case 2 classification still useful? Oceanography – Washingyon DC – Oceanography Society 17(2), 60–67.

Morel, A. & S. Bélanger (2006). Improved detection of turbid waters from ocean color sensors information. Remote Sensing of Environment 102(3), 237–249.

Morel, A. & L. Prieur (1977). Analysis of variations in ocean color1. Limnology and oceanography 22(4), 709–722.

Myint, S. & N. Walker (2002). Quantification of surface suspended sediments along a river dominated coast with noaa avhrr and seawifs measurements: Louisiana, usa. International Journal of Remote Sensing 23(16), 3229–3249. NASA (2003). Modis technical specifications. http://modis. gsfc.nasa.gov. NASA (2015). Viirs land. http://viirsland.gsfc.nasa.gov/ index.html. NASA (2016). Visible infrared imaging radiometer suite. http://npp.gsfc.nasa.gov/viirs.html. Nechad, B., K. Ruddick, & Y. Park (2010). Calibration and validation of a generic multisensor algorithm for mapping of total suspended matter in turbid waters. Remote Sensing of Environment 114(4), 854–866. Neukermans, G., K. Ruddick, E. Bernard, D. Ramon, B. Nechad, & P.-Y. Deschamps (2009). Mapping total suspended matter from geostationary satellites: a feasibility study with seviri in the southern north sea. Optics Express 17(16), 14029–14052. Neukermans, G., K. Ruddick, & N. Greenwood (2012). Diurnal variability of turbidity and light attenuation in the southern north sea from the seviri geostationary sensor. Remote Sensing of Environment 124, 564–580. Odermatt, D., A. Gitelson, V. E.

Brando, & M. Schaepman (2012). Review of constituent retrieval in optically deep and complex waters from satellite imagery. Remote sensing of environment 118, 116–126. Paola, C. &V.Voller (2005).A generalized exner equation for sediment mass balance. Journal of Geophysical Research: Earth Surface (2003–2012) 110(F4). Park, E. & E. M. Latrubesse (2014). Modeling suspended sediment distribution patterns of the amazon river using modis data. Remote Sensing of Environment 147, 232– 242. Pitarch, J., D. Odermatt, M. Kawka, & A. Wüest (2014). Retrieval of vertical particle concentration profiles by optical remote sensing: a model study. Optics express 22(103), A947–A959. Robinson, I. S. (2004). Measuring the oceans from space: the principles and methods of satellite oceanography. Springer Science & Business Media. Rosati, J.,T. Walton, & K. Bodge (2002). Longshore sediment transport. Coastal Engineering Manual (Part III), 2. Ruddick, K., C. Brockmann, R. Doerffer, Z. Lee,V. Brotas, N. Fomferra, S. Groom, H. Krasemann, V. Martinez-Vicente, C. Sa, et al. (2010). The coastcolour project regional algorithm round robin exercise. In SPIE Asia-Pacific Remote Sensing, pp. 785807–785807. International Society for Optics and Photonics. Staenz, K. & A. Held (2012). Summary of current and future terrestrial civilian hyperspectral spaceborne systems. In Geoscience and Remote Sensing Symposium (IGARSS), 2012 IEEE International, pp. 123–126. IEEE. Van Rijn, L. C. (1993). Principles of sediment transport in rivers, estuaries and coastal seas, Volume 1006. Aqua publications Amsterdam. Wang, J.-J. & X. Lu (2010). Estimation of suspended sediment concentrations using terra modis: An example from the lower yangtze river, china. Science of the Total Environment 408(5), 1131–1138. Woz´niak, S. B. & D. Stramski (2004). Modeling the optical properties of mineral particles suspended in seawater and their influence on ocean reflectance and chlorophyll estimation from remote sensing algorithms. Applied Optics 43(17), 3489–3503. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) ISBN 978-1-138-02977-4

A comparison of simple buoyant jet models with CFD analysis of overflow

dredging plumes

B. Decrop

Dredging and Offshore Department, IMDC, Antwerp, Belgium

T. De Mulder

Department of Civil Engineering, Ghent University, Ghent, Belgium

E. Toorman

Department of Civil Engineering, KULeuven, Leuven, Belgium

ABSTRACT: Trailing Suction Hopper Dredgers release excess water with a varying flow rate and with variable fine sediment content. In the recent past, the near-field dispersion of overflow dredging plumes was determined using simple integral solutions or Lagrangian models of the buoyant jet in cross flow. In reality, these negatively buoyant sediment plumes are interacting with the flow around the hull of the vessel, air bubbles and the propellers.

If these interactions are not taken into account for the near-field modelling, the source terms for far-field simulations of the environmental impact of turbidity are inaccurate. By consequence, the predictions of the environmental impact of the generated turbidity might not be accurate enough to avoid adverse effects later on in the project phase. In a CFD analysis in Ansys Fluent, it is investigated how these complex interactions take place and how they can be included in near-field dredging plume simulations. The CFD analysis reveals that the simple models can be relatively accurate in some cases, but that large deviations exist for most real-life situations.

1 INTRODUCTION

Overflow plumes are found when a Trailing Suction Hopper Dredger (TSHD) pumps benthic sediments along with a substantial fraction of seawater into its hopper (Figure 1). The lean mixture at the water sur face of the hopper contains a fraction of the dredged sediment and is discharged to improve dredging cycle efficiency. The mixture flows over the edge of the overflow cone or over the side of the vessel. Due to the balance of upward turbulent transport and settling of sediments, mainly fine sediments are found in the upper layers of the hopper content. Therefore, the turbidity of a dredging plume is mainly caused by fine sediments, with a typical increase in median grain size towards the end of a dredging cycle when the settled sand bed is closer to the hopper water surface. Models of hopper sedimentation have been developed in the past (van Rhee 2002). Many processes influence the sediment concentration and particle size distribution of the mixture released through the overflow: pumped material properties, the phase in the dredging cycle, hopper configuration, sloshing due to sea waves, the presence of air bubbles, et cetera. Once the water-sediment mixture is released, a negatively buoyant plume in a cross flow is formed in the waters below the TSHD, since the overflow is released

Figure 1. Situation sketch of a Trailing Suction Hopper Dredger working using the overflow shaft.

through the hull. The excess bulk mass density of the plume $\Delta\rho$ is defined as where ρ_∞ is the mass density of the sea water, ρ_m the mass density of the sediment-water mixture in the plume, c is the sediment mass concentration, ρ_s the mass density of the sediment material, φ_a is the volume fraction of air bubbles and ρ_a is the mass density of air bubbles. The phase in which the plume is mainly driven by the excess bulk mass density, is named the dynamic phase. At a certain distance from the overflow exit, dilution causes the sediment concentration to drop to a level at which the excess weight is no longer of impor tance against upward turbulent mixing. In this phase – referred to as the passive phase – the plume drifts with the background current and is called a passive plume. In case the dynamic plume does not entirely sink to the seabed at relatively short distance from the overflow exit, the mixed fine sediment forms a passive plume which can reach distances from the TSHD typically of the order of a few kilometer. In situ measurements of TSHD overflow dredging plumes are scarcely published, but some are available: Smith and Friedrichs (2011), Whiteside et al. (1995), CoastlineSurveys (1998), Spearman et al. (2011) and Breugem et al. (2009). These observations show that the plumes have a visible length of 1 to 2 km and have a lifetime of 30 to 60 minutes. In some circumstances the reduced translucency of the seawater and the deposition of sediments on the seabed can harm the environment. In the tender phase of dredging projects, increasingly strict environmen tal regulations require forecasting of dredging induced turbidity plumes, showing that while executing the planned dredging strategy, plumes will stay clear of the environmentally sensitive areas, or turbidity stays within the prescribed criteria. The prediction of the dispersion of dredging plumes becomes mandatory in more cases. Especially the dynamic phase plume sim ulation remains a challenge due to the complexity of the turbulent processes involved.These include a num ber of influencing factors such as air bubbles, ship hydrodynamics and propeller mixing. Large-scale modelling tools are capable to solve the passive part of the plume. However, in the modelling of dredging plumes the dynamic part of the plume is still a missing link between the sediment discharge at the overflow exit and the passive part of the plume. Today this gap is bridged by roughly estimating the sediment flux to the passive plumes. This is done by taking a fixed fraction of the sediments released through the overflow. In reality, there are reasons to believe this fraction can vary widely over time and space, depend ing on dredging speed, ambient flow velocity, overflow sediment concentration and air bubble entrainment in the overflow shaft. Indeed, a strong current can keep all released sediments in suspension, while a dense plume in a weak current can descend to the seabed with only a minor fraction of sediments remaining suspended in the water column (Decrop et al. 2014, de Wit et al. 2015). In the past, the transformation of the bulk over flow plume to a passive plume sediment source

has been executed with integral models representing

the integrated Navier-Stokes equations for a buoy

ant jet in crossflow (Fischer 1979, Jirka 2006). Such

models were implemented by e.g. Spearman et al.

(2011). However, this type of model cannot incor

porate the formation of a surface plume due to the

complex flow pattern around the vessel and due to air

bubbles. Table 1. Dimensionless distances and asymptotic solutions for the buoyant jet in crossflow, according to Fischer (1979). z M > z B z M < z B Abscissa ξ x z −1 M (C 1 /C 2) 6 x z B z −2 M (C 3 /C 1) 4 Ordinate ζ −z¯ z −1 M C 2 1 (C 2) −3 −z¯ z −3/2 M z 1/2 B C −3 1 C 2 3 κ (C 4 /C 1)(C 2 /C 1) (C 4 /C 3)(C 3 /C 1) 1/3 ξ c (critical ξ) κ 1/2 (z B /z M) 2 κ −3 (z M /z B) ξ 1 ζ = ξ 1/2 ζ = ξ 1/2 1 ξ ξ c ζ = ξ 1/3 ζ = ξ 3/4 ξ c ξ ζ = κ(z B /z M) 1/3 ξ 2/3 ζ = κ(z B /z M) 1/6 ξ 2/3 In this paper, it is shown from experimental results that the centerline of a free, undisturbed fine-sediment plume in a crossflow can be predicted by these integral laws for buoyant jets. Subsequently, a direct comparison is made between the simulations of overflow plumes executed by Decrop (2015) and some of the integral models found in literature. Finally, an analysis shows under which circumstances the integral models remain relatively accurate and under which circumstances integral solutions differ substantially from simulation results of a validated, realistic CFD. 2 INTEGRAL MODELS The asymptotic (integral) solutions from Fischer (1979) show that the horizontal and vertical coordinates of the plume centerline can be written in selfsimilar form by, respectively, ξ = x z −2 M z B (C 3 /C 1) 4 and ζ = − z¯ z −3/2 M z 1/2 B C −3 1 C 2 3 , where z¯ is the mean trajectory elevation, z M and z B are momentum and buoyancy length scales, C 1 = 1.8, C 2 = 1.44, C 3 = 1.8 and C 4 = 1.1, are constants determined by Wright (1977). This is valid in case z M < z B , i.e. when buoyancy is dominant before the crossflow bends the plume over. The full asymptotic solutions by Fischer (1979) are given in table 1, also for the case z B < z M , i.e. when momentum is dominant before the crossflow bends the plume over. A second theoretical approach was followed by Lee and

Chu (2003). In a Lagrangian framework, a diskshaped slice of the plume is followed from the ejection point onwards. The shape, position and orientation of the disk are changed during time-stepping according to the influences of buoyancy, crossflow and entrainment. In this way, both the centerline and half-width of a buoyant jet can be calculated, even in stratified conditions. In the present paper, this model has been applied along with the asymtotic solutions of Fischer (1979) in order to compare the plume properties from process-based CFD simulations with theoretical solutions. 3 SCALE MODEL RUNS Experimental data of sediment plumes in a crossflow have been acquired in a scale-model and analysed in

Figure 2. Comparison of experimental results with Lagrangian model by Lee and Chu (2003) and integral laws (Fischer 1979). Experimental centerlines in black diamonds, Lagrangian model centerlines grey line, integral laws in black line.

(Decrop 2015). Fine-sediment plumes with a wide variety in efflux and crossflow conditions have been released from a schematised vessel in a flume. The influence of the stern of a dredging vessel can be deter mined when comparing the plume trajectories with integral laws (Fischer 1979) or the Lagrangian model for buoyant jets in crossflow by Lee and Chu (2003). Likewise, a plume with trajectory far from the flow expansion and turbulent mixing induced by the stern would be expected to adhere closely to a prediction of a standard buoyant jet in crossflow. It was found that plumes are not influenced by the ship’s hull when the centerline (at the same distance as the stern) is located at a distance below the ship of more than two times the draft. In figure 2, a plume with relatively high jet flow (W_0) to crossflow velocity (U_0) ratio λ is shown ($\lambda = W_0/U_0 = 1.53$). It can be observed that the result ing plume has a centerline and upper extent sufficiently away from the hull, and is by consequence not influ enced by it. It can also be seen that the experimental plume trajectories show a tight fit with the theoretical buoyant jet predictions. In a plume with much stronger crossflow ($\lambda = W_0/U_0 = 0.69$) a trajectory close to the hull is found. It can here be seen in figure 3 that the plume is completely drawn upward by the stern sec tion. Centerline, upper and lower extents are located significantly higher than predicted by a simple buoy ant jet in crossflow model. In general, the Lagrangian model seems to be closer to the (start of the) plume trajectory compared to the integral model by Fischer (1979). Nevertheless, the integral solutions by Fis cher (1979) are very close to the Lagrangian model in most of the 36 investigated cases. For that reason, only the solutions by Fischer (1979) will be shown in the following. In figure 4, the asymptotic solutions by Fischer (1979) (in self-similar form, ζ) are plotted along with the non-dimensionalised measured trajectories,

$\hat{\zeta}$. From this analysis, it can be confirmed that the experimental plume trajectories with no influence of the stern are in line with the theoretical solutions. The application of integral laws for near-field dredging plume predictions seems therefore justified in case the plume is very similar to an undisturbed buoyant jet.

A comparison with the validated CFD model in the Figure 3. Same as previous figure, showing a plume with relatively stronger crossflow. Figure 4. Self-similarity solutions ζ of all observed sediment plumes in crossflow compared with non-dimensionalised measured plume trajectories $\hat{\zeta}$. Plumes of which the full trajectory is influenced by the stern drawn in black diamonds, plumes of which only the upper fringes are influenced in grey diamonds and plumes not influenced by the stern in black crosses. next section can subsequently reveal in which conditions the theoretical solutions remain valid for real-life overflow dredge plumes. 4 CFD SIMULATIONS A three-dimensional CFD model, describing the flows of water, sediment and air bubble phases in the near-field of an overflow dredging plume has been developed and described in earlier work (Decrop et al. 2014, Decrop 2015). The CFD model employs the Large-Eddy Simulation technique to resolve the larger scales of turbulent motions on the model grid (Leonard 1974). The dynamics of the water-sediment mixture are described using the mixture model (Ishii and Hibiki 2006), in which a single set of momentum equations is solved for the mixture. Additionally, the slip velocity of sediments relative to the water phase is determined. The motion of the air bubble phase is formulated as a discrete Lagrangian phase, for which the accelerations are determined from a force balance including gravity, pressure gradient, virtual mass and drag. The source of momentum and swirl produced by the propellers is simulated using an actuator disk approach. The model was validated in different steps, by

comparison with experiments and in situ measure

ments. Amongst other applications, the model was

used to evaluate the efficiency of an environmental

valve in reducing the sediment dispersion by avoiding the entrainment of air bubbles in the overflow shaft

(Decrop et al. 2015). The rationale of the development of this model, was

to provide more accurate near-field properties of the

sediment plumes, in order to define better source terms

in far-field plume dispersion modelling. In this paper,

the actual higher performance of the CFD model over

the classical theoretical solutions is demonstrated.

5 COMPARISON OF CFD AND THEORETICAL SOLUTIONS

A similar approach as for the comparison with experi

mental results is followed. First a number of cases show

the behaviour of a single plume as simulated using

the CFD model, by determining the plume centerline.

Subsequently, the theoretical approach is added for

comparison. As a synthesis, a large number of numer

ically simulated plume centerlines are transformed to

the self-similar form, after which they can be plotted

in one scatter plot along with the theoretical solutions. A first case is shown in figure 5. A plume with the

following boundary conditions was simulated using

the CFD model: initial sediment concentration C_0

was equal to 20 g/l, water depth H = 39 m, over

flow exit velocity W_0 = 3.2 m/s, ship speed-through

water U_0 = 1.5 m/s and the overflow shaft diameter

amounted to D = 1.1 m. The overflow was located at

72 m from the stern and a typical air bubble volume

fraction of 7% was included. The round markers indi cate the centerline of the plume as simulated by the CFD model, the full line is the asymptotic solution. It can be seen that in the initial phase of the plume, the centerlines start to diverge rapidly, after which a parallel path is followed. This can be attributed to the air bubbles, which act as a reduction of the bulk mass density of the plume in the early stages in the CFD model, but after the majority of the air bubbles has escaped, this effect disappears. Also, at x/D = 40, a limited effect of the stern and propeller suction is observed. A second effect of the air bubbles taken into account by the CFD model, is the formation of a surface plume due to the drag of rising air bubbles. This is a very important effect which cannot be accounted for using theoretical plume approximations, since the surface plumes have the potential to travel over long distances since they have very limited buoyancy. In a second case, exactly the same plume condi tions are implemented, but it is released from a ship with an overflow located at 20 m from the stern (Figure 6), rather than at 72 m. It is evident that the theoretical solution is identical, since the effect of vessel hydrody namics is not incorporated. The CFD results, however, differ substantially. This can be explained by the fact

Figure 5. Vertical slice along the axis of a plume, with

the CFD sediment concentration plotted in grey-scale ($\log_{10}(C/C_0)$), the CFD centerline in round markers and the theoretical solution in full line. Boundary conditions are C_0 = 20 g/l, H = 39 m, W_0 = 3.2 m/s, ship speed-through-water U_0 = 1.5 m/s and D = 1.1 m. The overflow was located at 72 m from the stern. Figure 6. Symbols as in figure 5. Plume is released from an overflow located at 20 m from the stern rather than at 72 m. Figure 7. Symbols as in figure 5. Keel clearance equal to 9 m, U_0 = 3 m/s, W_0 = 1.9 m/s. that the plume released from an overflow shaft shortly upstream of the stern is still close to the ship when it meets the influence of the flow divergence behind the stern and the propeller suction. In this case, the plume centerline from the CFD model is located more than 15m higher compared to the theoretical solution. If the theoretical solution would have been used as a near-field prediction, the elevation of the plume would be underpredicted. This would lead in turn to a far-field modelling underprediction of the capacity of the plume to travel long distances, potentially to environmentally sensitive areas. The effect of air bubbles and ship hydrodynamics becomes even more important when the velocity ratio is further reduced and the ships keel clearance is lower. In the third case shown in figure 7, W_0 = 1.9 m/s and U_0 = 3 m/s. Here, the theoretical solution predicts a plume descending to the sea bed after x/D = 68. When all relevant processes are taken into account using the CFD model, the air bubble-induced buoyancy takes over from the limited initial outflow momentum, leading to a plume clinging to the keel of the ship, attracted and mixed by the propellers. As a consequence, a surface plume is formed with a sediment concentration of about 1% of the overflow concentration, which can be a very high value from an environmental point of view. Only a limited number of cases can be found in which the theoretical solutions are reasonably valid in terms of plume centerline. Firstly when the plume has a high density and the crossflow is limited (e.g. when

Figure 8. Symbols as in figure 5. Keel clearance equal to

9 m, U_0 = 1.5 m/s, C_0 = 120 g/l, W_0 = 1.9 m/s.

Figure 9. Symbols as in figure 5. Keel clearance is 33.4 m,

U_0 = 1.5 m/s, W_0 = 3.2 m/s, C_0 = 20 g/l.

the ship is dredging along with the current or in shal

low water). In that case only the air bubbles have a limited effect on the nearly-vertical path of the plume (Figure 8). When all conditions are met for a plume with limited difference from a simple buoyant jet in crossflow (weak crossflow, overflow far from stern, high C_0, deep water) a plume is expected close to the theoretical solutions. When also the air bubbles are limited (as when using an environmental valve), the CFD model and theoretical solutions indeed match relatively well (figure 9). The only small deviation is due the remaining air bubbles ($\varphi_a = 0.007$). But even in this case the CFD model provides a wealth of additional information such as the mixing of sedi ments from the top of the plume by wake vortices and propellers. A large number of CFD simulations have been exe cuted by Decrop (2015). The results in this dataset of more than 100 cases are analysed in this paper to give an overview of (non-)adherence to theoretical solu tions. A similar analysis has been made as in figure 4. In order to assess the capability of a theoretical solu tion to capture the path of a dredging plume, the latter need to be classified in some way. Based on experimental results, plume trajectories were added to a diagram and exponential functions can be fitted forming the boundary between plume regimes (Decrop 2015). The primary variables to find a pattern in the plume behaviour were found to be the

Richardson number,

and the velocity ratio, A density current is found under the condition U_0 <

W_0 α_1 (Ri), while a horizontal plume is found in case

$U_0 > W_0 \alpha_2$ (Ri), with Figure 10. Theoretical solutions ζ_F of all simulated plumes compared in a scatter plot with non-dimensionalised plume trajectories of the CFD solutions ζ_N . Plumes classified as horizontal plumes drawn in black diamonds, transitional plumes in grey diamonds and plumes of type density current in black filled squares. where, $a_1 = 1.14$, $b_1 = 0.64$, $a_2 = 2.26$ and $b_2 = 0.81$. Transitional plumes are found in between those conditions. Here, the same classification is used to separate the full-scale overflow plumes in a scatter plot (figure 10). It shows for the different classes how well the Fischer (1979)-trajectories (ζ_F) adhere to the numerical CFD solutions (ζ_N). First observation is that all realistic plume solutions have a higher plume trajectory compared to the theoretical solutions (ζ is the non-dimensional form of distance below the ship keel). In an engineering context of dredging projects planning, this means theoretical solutions are not conservative, since the potential of plumes to travel long distances is underestimated. Second observation is the clear separation of the plume classes. Density current-type of plumes are relatively well described by the theoretical solutions. The horizontal type of plumes are not at all described well by the theoretical solutions. 6 CONCLUSIONS The validity of using theoretcal solutions for buoyant jets is investigated by comparison with a validated, highly detailed CFD model for overflow dredging plumes. It is found that in some specific cases of plumes which descend rapidly from the dredging vessel, these solutions are relatively accurate. However, in most other cases, there is a large discrepancy between the theoretical solutions and the realistic CFD solutions, certainly when air bubbles are entrained in the overflow shaft. In the latter cases, an important feature of overflow plumes is occurring: a surface plume which is detached from the main plume.

This feature is of great importance for the assess

ment and real-time forecasting of dredging plumes and

cannot be predicted by theoretical solutions. Given the fact that it cannot be a priori determined which type of plume will be encountered within the time and space-domain solution of plume dispersion in a far-field model, the theoretical solutions are not very adequate for near-field plume dispersion. For the determination of sediment source terms for far-field simulations, it is advised to apply either a CFD model of specific cases, or a parameterised model fitted using a set of CFD simulations.

Breugem, W., M. Bollen, M. Sas, & J. Vandenbroeck (2009). A field survey of a dredging plume during gravel mining. In CEDA Dredging Days 2009 – Dredging Tools for the Future, Rotterdam.

CoastlineSurveys (1998). Marine aggregate mining benthic and surface plume study. report prepared by arc marine ltd, south coast shipping company ltd, united marine dredging ltd and hr wallingford.

de Wit, L., C. van Rhee, & A. Talmon (2015). Influence of important near field processes on the source term of suspended sediments from a dredging plume caused by a trailing suction hopper dredger: the effect of dredging speed, propeller, overflow location and pulsing. Environmental Fluid Mechanics 15(1), 41–66.

Decrop, B. (2015). Numerical and Experimental Modelling of Near-Field Overflow Plumes. Ph. D. thesis, Ghent University, KULeuven.

Decrop, B., T. De Mulder, E. Toorman, & M. Sas (2015). Numerical simulation of near-field dredging plumes: Efficiency of an environmental valve. Journal of Environmental Engineering 141(12). Decrop, B., M. Sas, T. De Mulder, & E. Toorman (2014). Largeeddy simulations of a sediment-laden buoyant jet resulting from dredgers using overflow. In International Conference on Hydroscience and Engineering, pp. 737–744. Fischer, H. (1979). Mixing in inland and coastal waters. Academic Press. Ishii, M. & T. Hibiki (2006). Thermo-Fluid Dynamics. Springer. Jirka, G. H. (2006). Integral model for turbulent buoyant jets in unbounded stratified flows part 2: plane jet dynamics resulting from multiport diffuser jets. Environmental Fluid Mechanics 6(1), 43–100. Lee, J. & V. Chu (2003). Turbulent jets and plumes: A Lagrangian approach. Springer. Leonard, A. (1974). Energy cascade in large-eddy simulations of turbulent fluid flows. In Turbulent Diffusion in Environmental Pollution, Volume 1, pp. 237–248. Smith, S. & C. Friedrichs (2011). Size and settling velocities of cohesive flocs and suspended sediment aggregates in a trailing suction hopper dredge plume. Continental Shelf Research 31(10), S50–S63. Spearman, J., A. de Heer, S. Aarninkhof, & M. Van Koningsveld (2011). Validation of the TASS system for predicting the environmental effects of trailing suction hopper dredgers. Terra et Aqua (125). van Rhee, C. (2002). On the sedimentation process in a trailing suction hopper dredger. Ph. D. thesis, Delft University of Technology. Whiteside, P., K. Ooms, & G. Postma (1995). Generation and decay of sediment plumes from sand dredging overflow. In Proceedings of the 14thWorld Dredging Congress, Volume 14, pp. 877–892. Wright, S. J. (1977). Mean behavior of buoyant jets in a crossflow. Journal of the Hydraulics Division 103(5), 499–513.

Special session: Management of hydraulic systems by means of fuzzy logic

Fuzzy regression analysis between sediment transport rates and stream discharge in the case of two basins in northeastern Greece

M. Spiliotis

Department of Civil Engineering, Democritus University of Thrace, Xanthi, Greece

V. Kitsikoudis

Department of Civil Engineering, Istanbul Technical University, Istanbul, Turkey

V. Hrissanthou

Department of Civil Engineering, Democritus University of Thrace, Xanthi, Greece

ABSTRACT: Systematic measurements of sediment transport rates and stream discharge were conducted in two basins, in northeastern Greece. Separate measurements of bed load transport and suspended load transport were performed for these basins located near Xanthi (Thrace, northeastern Greece): Kimmeria Torrent basin with an area of about 35 km^2 and Kosynthos River basin with an area of about 237 km^2. Measured data of rainfall depth, rainfall duration, water discharge and sediment transport for the outlets of the above basins were available.

In this study, relationships between sediment transport rates andwater discharge are presented, based on nonlinear fuzzy regression, due to the fact that there is insufficient and no absolutely reliable data. Thus, two curves were studied regarding the basins of Kimmeria Torrent and Kosynthos River: (i) the suspended load transport rate versus rainfall intensity and water discharge and (ii) the

bed load transport rate versus water discharge. The selection of the fuzzy curves is proposed based on the aims to minimize the total fuzziness, while all the data must be included in the produced fuzzy band. However, overfitting behaviour must be avoided. Useful other conclusions are presented.

1 INTRODUCTION

The quantification of sediment transport in alluvial streams and rivers is necessary for studies span ning across diverse disciplines such as engineering, ecology, geomorphology, and biology. Nonetheless, while there have been significant developments in understanding the actual physics that govern sedi ment transport, at least in simple configurations, its quantification still poses a problem. The computation of sediment transport exhibits several difficulties and peculiarities, which inhibit the general applicability of analytical models and favour empirical formulae.

Sediment transport comprises the bed load, the sus pended load, and the wash load. Each one of these modes has a different driving mechanism.The bed load depends on the shear forces exerted on the bed by the flow, while the suspended sediment load is dictated by the turbulence intensity. The wash load consists of very fine particles and its transport depends on its upstream availability and not on the flow inten sity. The uncertainties in sediment transport prediction stem mainly from the role of turbulence on sediment entrainment (e.g. Sumer et al., 2003; Diplas et al., 2008) and suspended sediment transport (e.g. Ven ditti and Bennett, 2000), as well as from the irregular geometries of the streambed and the individual sediment grains (e.g. Fenton and Abbott, 1977; Kirchner et al., 1990). Such parameters are modeled statistically by taking into account the first and second statistical moments (i.e. the mean value and the standard deviation). However, the calibration of detailed deterministic sediment transport models brings additional complexity due to the increased data demand, which doesn’t always enhance the predictive accuracy (Barry et al., 2004). Performing that kind of measurements to acquire such data is a laborious and costly process. Moreover, detailed measurements in high flows are very difficult to obtain due to instrumentation and physical limitations. Empirical models provide a cheap and useful engineering tool, and have been utilized for numerous water related applications. More recently, data-driven modeling and soft computing techniques have exhibited superior performance when compared to the more traditional nonlinear regression in numerous hydrologic and hydraulic applications (Maier and Dandy, 2000). These techniques, with the inclusion of fuzzy logic, provide the means to handle uncertainty and generate models that can tolerate imprecision. Especially in sediment transport studies, it has been shown that they can omit the explicit determination of the critical flow conditions, which can be a source of error due to its vague nature, and lead to increased accuracy compared to the typical sediment transport equations (Kitsikoudis et al., 2014; 2015).

The present study employs fuzzy regression anal ysis for the quantification of bed load and suspended load, individually, with minimum data requirements. The former is modeled with respect to flow discharge, while the latter with respect to flow discharge and rainfall intensity. The generated fuzzy curves pro vide information about the upper and lower sediment transport limits, for specific values of the indepen dent variables, based on the utilized data uncertainty. This uncertainty and vagueness is inherent to turbu lent flows over an alluvial bed and, as a result, for the same independent variables the sediment transport rates may be different. This is mainly due to calcu lations based on timeand space-averaged variables, which are necessary simplifications for the applica bility of sediment transport formulae. In addition, the measurements that are employed for the formulae cal ibration inevitably contain some amount of noise and, to some degree, imprecision due to difficulties in sedi ment transport sampling (e.g. Helley-Smith samplers). Hence, the generated upper and lower sediment trans port limits pertain to differences in sediment transport rates due to flow non-uniformity, sediment properties, secondary flows, biota impact, sediment availability etc. The available measurements originate from two river basins in northeastern Greece.

In contrast to the statistical regression, the fuzzy regression analysis has no error term, while the uncer tainty is incorporated in the model by means of fuzzy numbers (Spiliotis and Bellos, 2015; Kitsikoudis et al., 2016).

The fuzzy regression may be a useful tool to express functional relationships between variables, especially when the available data are insufficient (Ganoulis, 1994). For example, Kitsikoudis et al. (2016) employed a fuzzy regression and sets to pro duce a lower and upper limit for the initiation of sediment motion, that is, a fuzzy band, which corre spond to weak sediment transport and general move ment, respectively. Thus, the ambiguity of selecting a threshold for the initiation of motion is avoided, and a smoother transition to the state of general movement is provided.

The data of the fuzzy regression can be either fuzzy or crisp. Usually, the data are rather crisp numbers, and thus, the uncertainty arises from the adopted fuzzy model.

2 FUZZY REGRESSION

The fuzzy regression analysis gives a fuzzy functional relationship between the dependent and independent variables (Papadopoulos and Sirpi, 1999). According to Tanaka et al. (1987) approach, the problem of fuzzy linear regression is finally formulated as an optimiza tion problem. In case that symmetrical fuzzy triangular numbers are used, the problem is transformed into Figure 1. Fuzzy triangular symmetrical number. a linear programming problem (Tanaka et al., 1982; Tsakiris et al., 2006). The fuzzy linear regression model proposed by Tanaka et al. (1987) and Tanaka et al. (1989) has the following form: with j = 1, . . ., m, i = 1, . . ., n where n is the number of independent variables, m is the number of data, and here, let $\tilde{A}_i = (a_i, c_i)_{TR}$ symmetric fuzzy triangular numbers selected as coefficients (Fig. 1), which have the membership function presented below: where a_i and c_i are the centres and the widths of the fuzzy coefficients, respectively (e.g. Klir and Yuan, 1995; Spiliotis et al., 2015). The model of fuzzy linear regression produces a fuzzy band, which can be calculated based on the extension principle of fuzzy sets and logic. In general, the extension principle enables us to define the crisp functions on a fuzzy domain and consequently the extension principle can be used in order to define the algebraic operations between fuzzy sets (e.g. Klir and Yuan, 1995). If the input data are crisp numbers, then the model of fuzzy linear regression can be interpreted mathematically by multiplying fuzzy numbers by crisp numbers, as well as by adding the fuzzy numbers. In case that the coefficients are fuzzy triangular numbers, the linearity remains also in the total regression output. The α-cut set of the fuzzy number A (with $0 < \alpha \leq 1$) is defined as follows: Notice that the α-cut set is a crisp set determined from the fuzzy set according to a selected value of the membership function and, alternatively, a fuzzy set can be practically derived from a significant number of α-cut sets. In case of α= 0, the above definition (Eq. 3) can be modified without the equality in order to describe the zero-cut (Kitsikoudis et al., 2016).

According to fuzzy linear regression model, the

center and the width of the fuzzy coefficients could

be determined by solving a constrained optimization

problem, whilst, in case of the classical regression, one

unconstrained optimization problem is solved. As it is

widely proposed, the objective function could be equal

to the total spread of the fuzzy outputs.

In case of focusing on the zero-cut and if fuzzy symmetrical numbers are selected as coefficients, it

holds:

where y^R, y^L is the right-hand boundary and the left

hand boundary, respectively, of the fuzzy set, which

are the boundaries of the zero-cut.

The constraints express the concept of inclusion of

the historical data within the produced fuzzy band.

The inclusion of a fuzzy setA to the fuzzy set B with

the associated degree $0 \leq a \leq 1$ is defined as follows

(Fig. 2):

In condition of fuzzy triangular numbers as coeffi

cients and by using the mentioned objective function,

the problem of fuzzy linear regression is reduced to a

linear programming problem. Indeed, based on the pre

vious description, the fuzzy linear regression analysis

is reduced to the estimation of $\tilde{A}_0$ and $\tilde{A}_i = (a_i, c_i)_{TR}$,

$i = 1, . . . , n$, that minimizes the spread of the fuzzy

output, subject to the inclusion constraints as follows

(Spiliotis and Bellos, 2015; Kitsikoudis et al., 2016):

subject to:

where $i = 0, 1, . . . , n$ and $j = 0, 1, . . . , m$.

In addition, $\sum_{i=0}^{n} a_i x_{ij} - (1 - \alpha) \sum_{i=0}^{n} c_i \lvert x_{ij} \rvert$, $\sum_{i=0}^{n} a_i x_{ij} +$

$(1 - \alpha) \sum_{i=0}^{n} c_i \lvert x_{ij} \rvert$ are the lower and the upper bound

aries, respectively, of the corresponding α-cut of $\tilde{Y}_j$.

As y_j, the jth observed data is meant, considering the dependent variable, which in this application is a crisp number.

It should be clarified that, if $\alpha \neq 0$ is selected, the fuzziness of the produced model is greater compared to $\alpha = 0$ (Papadopoulos and Sirpi, 1999). Many times, $\alpha = 0$ was selected though, so that the calculation pro

cedure can be simplified, in order to avoid a very large Figure 2. The concept of inclusion in the case of fuzzy regression. width. In addition, the optimal solution for $\alpha = 0$ can be easily achieved after considering the optimal solution of $\alpha = 0$, since the centre of Y_j remains the same, whereas the width can be evaluated using the width produced with $\alpha = 0$ (e.g. Papadopoulos and Sirpi, 1999). The presented approaches refer to fuzzy linear regression problem. The above methodology can be used to cover some cases of nonlinear regression. For instance, in case of polynomial form, a fuzzy linear regression formulation can be obtained by substituting some terms with fictitious auxiliary variables. In any case, a measure of the appropriateness of the proposed method is the value of the objective function. In general, successful models lead to small values of the objective function, which expresses the fuzzy spread. 3 CASE STUDIES 3.1 Kimmeria Torrent basin The basin of Kimmeria Torrent has an area of about 35 km^2 consisting of forest (55%), bushes (33%), urban area (1%) and an area with no significant vegetation (11%). The highest altitude of the basin is about 800 m. The length of the main stream of the basin is about 10 km. The mean soil slope gradient of the basin is about 45.5%, while the mean slope gradient of the main stream of the basin is about 6%. On certain days, water discharge, bed load transport and suspended load transport measurements, after rainfall events, were performed at the outlet of KimmeriaTorrent basin. Rainfall data (daily rainfall depth and rainfall duration) were available from the meteorological station of the Laboratory of Ecological Engineering and Technology (Department of Environmental Engineering, Democritus University of Thrace), located near the outlet of Kimmeria Torrent basin. The mean width of the cross sections, where the measurements were performed, was 7.0 m. In Table 1, the date of the measurements, the rainfall intensity, the water discharge, the suspended load transport rate and the bed load transport rate are presented. Sediment transport in the streams is classified into bed load transport and suspended load transport on

Table 1. Rainfall data and measured values of water discharge, suspended load and bed load transport rate at the outlet of Kimmeria Torrent basin.

Date	Rainfall intensity (mm/hr)	Water discharge (m^3/s)	Suspended load transport rate (g/s)	Bed load transport rate [kg/(s m)]
19-06-2004	2.80	0.67	403.2	0.0005
21-06-2004	36.80	3.05	2225.0	0.0008
22-06-2004	2.86	0.65	262.0	0.0005
23-06-2004	19.39	0.40	369.2	0.0005
30-06-2004	5.33	0.81	379.0	0.0006
22-06-2004	2.86	13.0	13.00	no measurem.
23-06-2004	19.39	0.59	757.9	no measurem.
20-05-2005	3.85	0.23	3.58	no measurem.
13-06-2005	13.94	0.20	7.04	no measurem.
21-09-2007	2.56	0.04	0.004	0.00008
20-11-2007	–	0.06	0.07	0.000014
20-11-2007	2.58	1.26	15.28	0.042
11-12-2007	5.68	0.805	9.52	0.1013
06-04-2008	3.67	3.09	148.6	0.2156

the basis of the two different kinds of motion. The bed load comprises coarse material that is entrained from the bed, and depends on the hydraulic conditions. The suspended sediment load consists of finer bed material as well as fine sediment emanating from the rainfall induced soil erosion, thus it additionally depends on rainfall characteristics (Metallinos and Hrissanthou, 2010).

On the basis of the above thoughts, the follow ing fuzzy nonlinear regression relationships were established for the basin of Kimmeria Torrent:

- Suspended load transport rate versus rainfall inten sity and water discharge.
- Bed load transport rate versus water discharge.

As mentioned before, every fuzzy regression curve is produced by following a constrained optimization process (Eqs. 6 and 7). The objective function repre sents the fuzziness of the model and the constraints represent the requirement that all the data must be included in the fuzzy band for the selected α-cut (here $\alpha = 0$, as it is widely used).

In case of the suspended load transport rate, the fol lowing fuzzy nonlinear relation between the suspended load transport rate, m_s, versus the water discharge, Q, and rainfall intensity, r, was obtained (Fig. 3):

where j refers to the data and m is the total number of data.

An interesting perspective is that only two terms have a fuzzy spread, that is, the remaining coefficients have no uncertainty, and hence they are conventional

numbers. The objective function has the following value: where m R s,j , m L s,j the right-hand and the left-hand boundary (from the zero-cut), respectively, of the produced fuzzy suspended load transport rate. It is obvious from Figure 3 that, for a considerable range of low rainfall intensity, the suspended load transport rate is independent of the water discharge. This is due to the fact that runoff and soil erosion in the corresponding basin are quantitatively low, so that suspended load transport is not influenced significantly by the runoff in the main stream of the basin, as well as by soil erosion in the basin considered. Another interesting point of view, is that, by applying the conventional regression analysis with the same form as the fuzzy model, we are led to irrational overfitting.The conventional regression analysis is marked with yellow color in Figure 3. If a polynomial of higher degree is tested, then a small improvement of the objective function occurs. However, by using higher degree polynomial, the danger of overfitting can appear, especially in case that there is not enough data. Indeed, this occurs by using a cubic polynomial fuzzy regression, as it will be explained below. For illustrative purpose, the cubic polynomial fuzzy relation (Fig. 4) and its objective function are presented below: Then, the objective function has the following value: A determinant criterion in order to select the model, is the value of the fuzzy spread without overfitting. If only one independent variable is selected, a significant larger value of the fuzzy spread is produced and hence, this model is discarded. In case of more complex relations, a small improvement of the objective function is achieved and hence, this model is also rejected. On the other hand, if a cubic polynomial is selected, the fuzzy band is reduced. However, an overfitting behaviour is clear in Figure 4. Indeed, extremely large irrational values for the suspended load transport rate appear and furthermore the monotony between the suspended load transport rate and the rainfall intensity does not remain as in Figure 3. Thus, sometimes, a low value of the objective function could hide an overfitting behaviour. Another interesting point of view is that, if only one independent variable is taken into account, the objective function, that expresses the total fuzziness, increases significantly.

Figure 3. Graphical representation of the suspended load transport rate as a fuzzy quadratic polynomial function of rainfall

intensity and water discharge, for Kimmeria Torrent. The conventional regression is depicted with yellow color.

That is, the fuzzy band, many times, incorporates parameters, which is impossible to take into account, either because of unawareness (e.g. due to the problem complexity, no reliable measurements etc.) or because of lack of data.

By the same way, a nonlinear fuzzy relation is pro duced for the bed load transport rate, m G , with respect to the water discharge, Q. Based on the minimization of the fuzzy band and in order to avoid the overfitting, a fuzzy quadratic polynomial is proposed, as follows (Fig. 5):

The semi-sum of the fuzzy band for all the data is equal to: If a fuzzy cubic polynomial relation is applied, the uncertainty appears only in the constant term: Indeed, a small reduction of the fuzzy band is achieved, if the polynomial of third degree is used: As it can be seen from Figure 6, in case of a fuzzy cubic polynomial regression, an overfitting curve results, since the monotony significantly changes between the points of data. By applying the conventional quadratic regression, the following curve is achieved: From Figure 5 it is evident that at least for the range of the water discharge, based on the existent data, it is

Figure 4. Graphical representation of the suspended load transport rate as a fuzzy polynomial, of third degree, func tion of rainfall intensity and water discharge, for Kimmeria Torrent.

Figure 5. Graphical representation of the nonlinear (quadratic polynomial) fuzzy regression for the bed load

transport rate [kg/(s m)] with respect to the water discharge (m^3/s). The black curve represents the conventional quadratic regression.

Figure 6. Graphical representation of the nonlinear (cubic polynomial) fuzzy regression for the bed load transport rate [kg/(s m)] with respect to the water discharge (m^3/s). The overfitting behaviour is obvious.

rather no safe choice to use the conventional quadratic regression. In fact, a band for the bed load transport rate seems a more reasonable prediction than a crisp value.

3.2 Kosynthos River basin

The basin of Kosynthos River has an area of about 237 km^2 consisting of forest (74%), bushes (4.5%), urban area (1.5%) and an area with no significant vegetation (20%). The highest altitude of the basin is about 1700 m. The length of the main stream of the basin is about 35 km. The whole basin can be divided into ten natural sub-basins with areas between 16 and 35 km^2. The mean soil slope gradient of the sub-basins

Table 2. Rainfall data and measured values of water discharge, suspended load and bed load transport rate near the outlet of Kosynthos River basin.

Date	Rainfall intensity (mm/hr)	Water discharge (m^3/s)	Suspended load transport rate (g/s)	Bed load transport rate [kg/(s m)]
26-10-2005	–	0.43	0.37	0.0018
02-11-2005	–	0.43	0.007	0.0021
30-11-2005	–	2.79	26.80	0.0031
07-12-2005	–	2.74	12.00	0.0045
14-12-2005	–	0.99	11.80	0.0034
26-03-2006	–			

5.99 no measurem. 0.0035 08-04-2006 3.34 3.20 no measurem. 0.0044 20-04-2006 4.50 2.68 no measurem. 0.0070 01-05-2007 3.45 2.24 678.9 0.0033 02-05-2007 2.80 2.89 1976.2 0.0040 03-05-2007 2.17 3.43 2336.2 0.0042 04-05-2007 1.60 2.44 1383.0 0.0035 11-05-2007 3.60 1.65 455.5 0.0030 is about 37%, while the mean slope gradient of the main streams of the sub-basins is about 5%. On certain days, measurements of water discharge, suspended load transport and bed load transport, after rainfall events, were performed near the outlet of Kosynthos River basin. Rainfall data (daily rainfall depth and rainfall duration) were available from the meteorological station of the Laboratory of Hydrology and Hydraulic Structures (Department of Civil Engineering, Democritus University of Thrace), located in the centre of gravity (Oreo) of the basin. In Table 2, the date of the measurements, the rainfall intensity, the water discharge, the suspended load transport rate and the bed load transport rate are presented. The use of a fuzzy linear logarithmic relation between the bed load transport rate, m G , and the water discharge, Q, was first examined (Fig. 7): The corresponding fuzzy band of the data is equal to: Then, the use of a fuzzy nonlinear logarithmic relation between the bed load transport rate, m G , and the water discharge, Q, was examined. The following

Figure 7. Graphical representation of logarithmic linear

fuzzy regression for the bed load transport rate [kg/(s m)]

with respect to the water discharge (m 3 /s), for Kosynthos

River. The black line represents the conventional logarithmic

linear regression.

Figure 8. Graphical representation of logarithmic nonlin

ear (quadratic polynomial) fuzzy regression for the bed load

transport rate [kg/(s m)] with respect to the water discharge

(m 3 /s), for Kosynthos River. The black curve represents the

conventional quadratic polynomial regression.

logarithmic quadratic polynomial curve was obtained

(Fig. 8):

The corresponding fuzzy band is equal to:

An interesting point of view is that by applying the

conventional logarithmic linear regression, an under

estimation of the bed load transport rate is concluded.

Furthermore, in case of the conventional logarithmic

quadratic polynomial regression model, there is an

interval where the bed load transport rate is reduced

when the discharge increases, which is a rather irra

tional conclusion. In case that there is not enough

observed data available, the conventional regression

can lead more easily to such irrational outcomes. Figure 9. Graphical representation of the nonlinear (quadratic polynomial) fuzzy regression for the bed load transport rate [kg/(s m)] with respect to the water discharge (m^3/s), for Kosynthos River. Subsequently, the use of a fuzzy quadratic polynomial regression was examined. The following fuzzy relation was obtained (Fig. 9): The corresponding fuzzy band is equal to: However, in case of the fuzzy quadratic polynomial, a significant overfitting trend can be observed. The authors suggest that in this case an overfitting behaviour holds, since for high values of the water discharge a significant reduction of the bed load transport rate seems to occur, which is without physical meaning. Thus, regarding the Kosynthos River basin, the logarithmic relation suits better to relate the bed load transport rate with the water discharge. Additionally, in this basin, the data were not enough to take a curve for the suspended load transport. As mentioned before, the fuzzy regression produces an upper and a lower boundary in which all the data must be included. In the problem of sediment transport, it seems more suitable to have a fuzzy band instead of a simple curve due to the complexity of the phenomenon. On the other hand, even if the conventional methods can be expanded to produce a confidence interval, this interval has not the ability to include all the data, as in case of the fuzzy model. 4 CONCLUSIONS Instead of the well known regression, a fuzzy nonlinear regression is successfully used to determine a fuzzy relation regarding the suspended and the bed load transport rate. The implementation of the proposed fuzzy regression method, many times, leads to curves in which

the coefficients of the independent variables have no

uncertainty.

The first criterion of the successful implementation

of the proposed method is a rational value of the sum

of the fuzzy band with respect to all data. However,

a complex relation could lead to curves with overfit

ting behaviour as in case of cubic fuzzy polynomial,

especially if there is no enough data.

Another interesting point of view is that, if only

one independent variable is taken into account, regard

ing the estimate of suspended load transport rate, the

objective function, that expresses the total fuzziness,

increases significantly. That is, the fuzzy band, many

times, incorporates parameters, which is impossible

to take into account, either because of unaware

ness (e.g. due to the problem complexity, no reliable

measuremnets etc.) or because of lack of data.

Regarding the bed load transport rate, it can suc

cessfully be described either by using a quadratic

polynomial fuzzy regression, or by using a loga

rithmic polynomial regression, with respect to water

discharge.

Finally, compared with the fuzzy regression analy sis, the conventional regression analysis can lead more easily to an overfitting scheme, especially when two independent variables exist. In most cases, the conven tional analysis, simply, can not address the simulation process. In fact, a band for the sediment load trans port rate seems a more reasonable prediction than a crisp value, as the conventional regression does. In any case, if there is not enough observed data, the fuzzy regression is a better solution compared with the conventional regression analysis.

Angelis, I., Metallinos, A. and Hrissanthou, V., 2012. Regres sion analysis between sediment transport rates and stream discharge for the Nestos River, Greece. Global NEST Journal, Vol. 14(3): 362-370.

Barry, J.J., Buffington, J.M. and King, J.G., 2004. A gen eral power equation for predicting bed load transport rates in gravel bed rivers. Water Resources Research 40(10): W10401.

Diplas, P., Dancey, C.L., Celik, A.O., Valyrakis, M., Greer, K. and Akar, T., 2008. The role of impulse on the initia tion of particle movement under turbulent flow conditions. Science 322(5902): 717-720.

Fenton, J.D. and Abbott, J.E., 1977. Initial movement of grains on a stream bed: the effect of relative protrusion. Proceedings of the Royal Society A 352(1671): 523-537.

Ganoulis, J., 2008. Engineering Risk Analysis of Water

Pollution: Probabilities and Fuzzy Sets. Wiley-Blackwell. Kirchner, J.W., Dietrich, W.E., Iseya, F. and Ikeda, H., 1990. The variability of critical shear stress, friction angle, and grain protrusion in water-worked sediments. Sedimentology 37(4): 647–672. Kitsikoudis, V., Sidiropoulos, E. and Hrissanthou, V., 2014. Machine learning utilization for bed load transport in gravel-bed rivers. Water Resources Management 28(11): 3727–3743. Kitsikoudis, V., Sidiropoulos, E. and Hrissanthou, V., 2015. Assessment of sediment transport approaches for sandbed rivers by means of machine learning. Hydrological Sciences Journal 60(9): 1566–1586. Kitsikoudis,V., Spiliotis, M. and Hrissanthou,V., 2016. Fuzzy regression analysis for sediment incipient motion under turbulent flow conditions. Environmental Processes, in press (doi:10.1007/s40710-016-0154-2). Klir, G. and Yuan, B.T., 1995. Fuzzy Sets and Fuzzy Logic Theory and its Applications. Prentice Hall. Maier, H.R. and Dandy, G.C., 2000. Neural networks for the prediction and forecasting of water resources variables: a review of modeling issues and applications. Environmental Modelling and Software 15(1): 101–124. Metallinos,A. and HrissanthouV., 2010. Regression relationships between sediment yield and hydraulic and rainfall characteristics for two basins in northeastern Greece. In Environmental Hydraulics, Two Volume Set: Proceedings of the 6th International Symposium on Enviornmental Hydraulics, Christodoulou G., Stamou A. (Eds), Athens, Greece, 23–25 June 2010. Papadopoulos, B. and Sirpi, M., 1999. Similarities in fuzzy regression models. Journal of Optimization Theory and Applications 102(2): 373–383. Spiliotis M., Garrote L. and Chavez-Jimenez A., 2015. Reorganization of water demand under changing conditions with possibilistic programming. Journal of Hydroinformatics 17(2): 239–259 (doi: 10.2166/hydro. 2014.008), (IWA Publishing, in press). Spiliotis, M. and Bellos, C., 2015. Flooding risk assessment in mountain rivers. EWRA 9th World Congress, Water Resources Management in a Changing World: Challenges and Opportunities, Istanbul, 10–13 June 2015. Sumer, B.M., Chua, L.H.C., Cheng, N.-S. and Fredsoe, J., 2003. Influence of turbulence on bed load sediment transport. Journal of Hydraulic Engineering 129(8): 585–596. Tanaka, H., 1987. Fuzzy data analysis by possibilistic linear models. Fuzzy Sets and Systems 24: 363–375. Tanaka, H., Hayashi, I. and Watada, J., 1989. Possibilistic linear regression analysis for fuzzy data. European Journal of Operation Research 40(3): 389–396. Tsakiris, G., Tigkas, D. and Spiliotis, M.,

2006. Assessment of interconnection between two adjacent watersheds using deterministic and fuzzy approaches. European Water 15/16: 15–22. Venditti, J.G. and Bennett, S.J., 2000. Spectral analysis of turbulent flow and suspended sediment transport over fixed dunes. Journal of Geophysical Research 105(C9): 22035–22047. Sustainable Hydraulics in the Era of Global Change – Erpicum et al. (Eds.) © 2016 Taylor & Francis Group, London, ISBN 978-1-138-02977-4

Rainfall data regression model using fuzzy set logic

Christos Tzimopoulos & Chris Evangelides

Aristotle University of Thessaloniki, Greece

Basilios Papadopoulos

Democritus University of Thrace, Xanthi, Greece

ABSTRACT: Classical linear regression has been used to measure the relationship between rainfall data in

different meteorological stations, in order to evaluate a linear relation and to predict the values of rainfall in

one station (dependent variables), from the rainfall values of an other station (independent variables). Classical

linear regression makes rigid assumptions about the statistical properties of the model, accepting the error

terms as random variables, and the violation of this assumption could affect the validity of the classical linear

regression. Fuzzy regression assumes ambiguous, imprecise parameters and data and may be more effective than

classical regression. In this paper, we evaluate the relation between rainfall data of Aggistron and Ano Vrontou

meteorological stations, which are located in the region of Central Macedonia (Northern Greece), using fuzzy

regression. The proposed model is a slight modification of Tanaka-Ishibuchi model with quadratic membership

function and interactive fuzzy parameters. In this model, the dependent observed rainfall values are crisp, and the

independent observed rainfall values as well as the parameters of the model are fuzzy. The results are presented

with two credibility degrees $h = 0$ and $h = 0.5$.

1 INTRODUCTION

Rainfall measurement models have been extensively used in the design process of water resources projects such as hydrological prediction, spillway design, cli matic change studies, rainfall and runoff correlation etc. Rainfall measurements in a specific area are com monly displayed in the form of time series where recorded values can be either continuous or discrete. In many instances, there is a correlation between rainfall time series that belong to different stations and com prise measurements with differing range. E.g., there is an available time series of 15 years for station A and a time series of 30 years for station B. Due to the corre lation between them, we can fill in the missing values, in order to extend the shorter time series.

Correlation analysis is used to depict the relation between the dependent variable (usually the mete orological station with the shortest data-recording time span) and the independent variables (neighbour ing stations with long recording time span). For this correlation, a multiple linear regression model is used:

In classical linear regression, the difference ε i between measurement values and estimated values, is a random variable with normal distribution and is considered to be caused by measurement errors. According to this, classical regression is considered to be probabilistic and has many uses but can be rendered problematic if the data set is small, if it's hard to prove that error distribution is normal, if there is fuzziness between dependent and independent variables or if linearity acceptance is not proper. Nowadays, new regression models have been introduced based on fuzzy logic (Tanaka et al, 1982; Tanaka, 1987; Tanaka and Watada, 1988; Tanaka and Hayashi, 1989; Tanaka and Ishibuchi, 1991; Papadopoulos and Sirpi, 1999; 2004; Tzimopoulos and Papadopoulos, 2013). In fuzzy regression the difference between measurement values and estimated values is attributed to the inherent fuzziness of the system as well as to the fuzziness of input and output data. In contrast with classical regression analysis, fuzzy regression analysis uses fuzzy functions for the regression factors. The above problem (Tanaka and Hayashi, 1989; Redden and Woodall, 1996) usually meets one of the three cases: a) crisp input values x_{ij} and output values y_j b) crisp input values x_{ij} and fuzzy output values $\tilde{y}_i$ c) fuzzy input values $\tilde{x}_{ij}$ and fuzzy output values $\tilde{y}_i$. In all of these cases, estimated values $\tilde{Y}_i$ are fuzzy. The adjustment of a fuzzy regression model can be achieved through two general methods: a) The possibilistic model (Tanaka et al, 1982;Tanaka, 1987; Redden and Woodall, 1996; Savic and

Pedrycz, 1991, etc.). Fuzzy regression is possi

bilistic and the membership function $\mu_{\tilde{A}}(x)$ of a

fuzzy number $\tilde{A}$ is considered equal to the pos

sibility distribution function $\pi_x(x)$. The fuzziness

of the model is minimized by taking into account

the minimum of the spreads around the centre of

the fuzzy parameters, while considering that the

experimental values of every sample are within a

specific interval of possible values. It is to point out

that possibilistic parameters in the models are non interactive, i.e. the joint possibilistic distribution of parameters is defined by minimum operators.

b) The least squares model (Diamond, 1990; Chang and Lee, 1996; Yang and Liu, 2003). The distance between the estimated output value of the model Y˜ i and the observed output value y˜ i is minimized. This method of Diamond is considered to be an exten sion of the classical linear regression method, based on the notion of model efficiency optimization depending on data.

In this article, quadratic membership functions as defined by Celmin¸ š (1987) and Tanaka and Ishibuchi (1991) are considered to propose a method of interac tive fuzzy parameters in possibilistic linear hydrolog ical systems, located in the region of Central Macedo nia (Northern Greece), and the method can be reduced to linear programming.

Celmin¸š wanted to maximize the membership val ues of the observations by minimizing the sum of squares of the deviations of the membership values from one. He obtained relatively simple algorithms if the membership functions of the data vectors belonged to a particular class of conical functions. Tanaka and Ishibuchi used a method similar to one considered by Celmin¸š, but the proposed approach is simpler and more understandable than Celmin¸š approach. They used quadratic membership functions as defined by Celmin¸š, and proposed an identification method of interactive fuzzy parameters in possibilistic linear systems.

Here we use the proposed method by Tanaka and Ishibuchi with a modification in the membership function of observed data, in order to obtain inter val inclusion between measured and estimated values. Input measured data were considered crisp (rainfall measurement station Aggistro) and output values were considered fuzzy (rainfall measurement station Upper Vrontou). Triangular membership functions were used for measured output values.

2 MATHEMATIC MODEL

2.1 Definitions

Definition 1. By setting a circumflex on a capital or lower case letter we define the number as fuzzy. Thus, $\tilde{K}$, $\tilde{C}$, $\tilde{A}$, $\tilde{y}$ etc. denotes fuzzy numbers.

Definition 2. Fuzzy set $\tilde{A}$, subset of a set X, is a function of the form: $\tilde{A}: x \in X \to \tilde{A}(x) \in [0,1]$, where

$\tilde{A}(x) = \mu_{\tilde{A}}(x)$ is its membership function. Definition 3. A triangular fuzzy number is defined as $\tilde{A} = (m, c_1, c_2)$, where m = the point of the triangle base where $\mu_{\tilde{A}}(m) = 1$, c_1 = the spread of the base on the left side of m and c_2 = the spread of the base on the right side of m. Definition 4. This number has the following properties: i. The membership function $\mu_{\tilde{A}}(x)$ is equal to 0 in the space $(-\infty, m - c_1]$. ii. Monotonously increasing in $[m - c_1, m]$.

iii. Equal to 1 in point m. iv. Monotonously decreasing in the space [m, m + c 1]. v. Equal to 0 in the space [m + c 1 , +∞) Definition 5. A symmetric triangular fuzzy number is defined as Ã = (m,c). Definition 6. An α-cut of Ã is written as [Ã] α and defined as {x|μ Ã (x)≥ α}for 0 < α≤ 1. [Ã] 0 is defined as the closure of the union of all [Ã] α , 0 < α≤ 1. Since the α-cuts of fuzzy numbers are always closed bounded intervals, then: [Ã] α = [A " (α), A r (α)], for every α. 2.2 Model development Consider a fuzzy dependent variable Y˜ j and x ij the independent variables influencing the variable Y˜ j . The result of fuzzy linear regression is an equation of the form: (Tanaka et al, 1982; Tanaka, 1987; Tanaka and Watada, 1988; Tanaka and Hayashi, 1989; Tanaka and Ishibuchi, 1991, etc.), where the measured input values x ij are crisp numbers and the measured output values y˜ i are fuzzy numbers. The parameters Ã i = (r i , c i) are considered symmetrical triangular fuzzy numbers. The elements r i , c i are respectively the mean and the spread of the parameter Ã i . The membership functions are usually written with the help of L, R numbers and have the property: L(0) = R(0) = 1, L(1) = R(1) = 0, and L −1 (h) = R −1 (h) = 1 − h. The confidence h-cuts (αcuts), (Chang and Lee, 1996), of the parameter Ã i are given as follows: and if they are multiplied with x ij , the following will result: Because the confidence h-cuts behave as intervals (Moore, 1966, Kaufmann and Gupta, 1991), the

above sum (1), which is a sum of many intervals, will

become:

or

Tanaka and Ishibuchi (1991) propose for the mem

bership function of Y˜ the following expression:

and they prove it as equal to:

According to their theory, the following steps are

followed:

1) Given the input-output data (x i ,y˜ i , i = 1, . . . ,n) and

a threshold h, it must hold an inclusion:

where [Y˜ ∗ j] h is a h-level defined by:

2) The following sum of spreads of the estimated fuzzy

[Y˜ * j], i = 1, . . . ,n, should be:

where the matrix C is proved to be a positive semi

definite matrix.

The problem now is formulated as follows:

where:

and the membership function of the output data y˜ i is

defined as: The solution of this problem according to Tanaka and Ishibuchi is as follows: 1st phase An optimum vector r * is found, that minimizes the expression: This solution constitutes the classical linear regression solution. 2nd phase The following optimization problem is solved with linear programming: This problem is called min problem according to Tanaka and Ishibuchi. If the optimum solution C * is a positive semi-definite matrix then, (r * , C *) is the solution of the problem. Otherwise, the third phase follows: 3rd phase The following orthogonal constraints are added to the above problem: and the problem is solved including these conditions. The solution is (r * , C *). In relation (10), I is the set of subscripts of the independent vectors {I = (1, 2, ...n)}. 3 APPLICATION We consider the rainfall measurement stations of Aggistro and Ano Vrontou with the following data: For this case, Equation (1) becomes: Y˜ j = Ã 0 + Ã 1 x 1j , x 0j = 1. Utilizing classical statistics the vector r * = (-11.941, 1.8181) is produced. The problem now is formulated as follows: where the numbers (154, 271, …., 557, 47) mean max(k 1 , k 2) 2 and k 1 , k 2 are:

Table 1. Mean monthly rainfalls – Crisp inputfuzzy output

data. Ano

T	x i	Aggistro-X	Vrontou-Y	e
1929-1930	x 1	47.60(mm)	65.80(mm)	13.16
1930-1931	x 2	65.60	118.90	23.78
1931-1932	x 3	40.30	57.00	11.40
1932-1933	x 4	30.80	52.50	10.50

1933-1934 x 5 45.10 61.10 12.22

1934-1935 x 6 54.10 73.90 14.78

1935-1936 x 7 58.70 104.40 20.88

1936-1937 x 8 60.60 83.10 16.62

1937-1938 x 9 55.10 78.80 15.76

1938-1939 x 10 45.90 79.00 15.80

1939-1940 x 11 54.70 106.50 21.30

1940-1941 x 12 47.50 77.50 15.50

Table 2. The optimal value of J vs. x i , x j .

Combination x 1 , x 12 x 1 , x 2 x 1 , x 3 x 1 , x 4

Optimal J 22717852 12806 15743 5822

Combination x 1 , x 5 x 2 , x 3 x 2 , x 4

Optimal J 77985 6627 5817

The following matrix is produced from the solution:

This matrix is not positive semi-definite and so the

following restriction is added to the problem:

The new matrix that is produced, is:

This matrix is positive semi-definite. Table 2 shows

the values of J, for various combinations of x i , x j vec

tors, while Figure 1 the variation of y versus x for h = 0

and h = 0.5.

Remark: As shown in Figure 1, the supports for

h=0 do not include the observed rain measurements of

the Ano Vrontou station with their deviations, in the

entire study region (points y±e). The inclusion con tains only the points y±e 1/2 . This result is due to the

fact that Tanaka-Ishibuchi used as membership func

tion of the data y: $\mu_{Y_i}(y) = \max\{1 - (y - y_i)^2/e_i, 0\}$.

In order, for the inclusion, to contain the points y±e,

the membership function is modified and takes the

following form: $\mu_{Y_i}(y) = \max\{1 - |y - y_i| e_i, 0\}$. Figure 1. The estimated fuzzy output and the given data, where: ▮= $y_i \pm e^{1/2}_i$, ©= $y_i \pm e_i$. From the new solution the vector r^* is obtained: The problem now becomes: The following matrix is produced from the solution: This matrix is not positive semi-definite and so the following restriction is added: The new matrix produced is: The above matrix is positive semi-definite and is the solution to the problem. Table 3 shows the values of J, for various combinations of x_i, x_j vectors, while Figure 2 shows the variation of y versus x for h = 0 and h = 0.5. As shown in Figure 2 the support for h=0 includes fully the observed rain measurements of theAnoVrontou station with their deviations, in the entire study region (points y±e).

Table 3. The optimal value of J vs. x_i, x_j.

Combination	Optimal value J
x_1, x_{12}	92180618
x_1, x_2	37420.022
x_1, x_3	46241
x_1, x_4	16878
x_1, x_5	294186
x_2, x_3	19639
x_2, x_4	16863.97
x_4, x_1	16878
x_4, x_2	16863
x_4, x_3	24507

x_4, x_5 16883

x_4, x_7 16866

Figure 2. The estimated fuzzy output and the given data

using the new modified method.

Figure 3. Membership functions (original and modified) of

measured data at x = 58.70.

4 CONCLUSIONS

In this paper, the Tanaka and Ishibuchi approach is

considered, which could identify interactive fuzzy

parameters in a possibilistic linear model of rainfall

station measurements.

The original Tanaka and Ishibuchi approach does

not insure data inclusion inside estimated supports,

and for that reason a modified version of Tanaka and

Ishibuchi approach is proposed here, which insures

Optimal spatial allocation of groundwater under fuzzy hydraulic parameters

E. Sidiropoulos

Aristotle University, Thessaloniki, Greece

ABSTRACT: A problem of optimally allocating groundwater is presented, that involves extraction and trans

portation cost. The groundwater model that underlies the allocation process presents an uncertainty in its basic

physical parameter, namely the hydraulic conductivity. This uncertainty is expressed by means of fuzzy sets. The

hydraulic conductivity is not a decision variable in the optimization problem. However, it renders fuzzy the cost

function to be minimized. Therefore, in the present problem

the objective function is an appropriate measure of

the fuzzy cost function. The concept of α-cuts is utilized for the computation of this function and of its measure.

The optimization problem is treated by means of a special genetic algorithm, the operational genetic algorithm

(OGA), which has been introduced elsewhere for the solution of spatial problems with crisp objective functions.

A discussion is carried out concerning features of the resulting spatial configurations, especially in terms of

compactness. Differences between crisp and fuzzy results are considered and discussed.

1 INTRODUCTION

The present paper deals with a combined problem of water extraction and water allocation. The extraction of water involves groundwater pumping and aquifer exploitation, while the allocation introduces a spatial character into the whole process. The spatial aspect dominates and brings the problem to the area of spa tial optimization. Related and sometimes more general problems of this category concern resource allocation and land use design.A genetic algorithm geared toward spatial problems has been presented by the author and utilized for related applications (e,g. Sidiropoulos and Fotakis, 2009 and 2011). This algorithm, called the operational genetic algorithm (OGA) is based on cel lular concepts for local selection of attributes and for global emerging characteristics.

The need for adding fuzzy aspects to the treatment of the problems just mentioned is self-evident. Indeed, a lot of fuzzy analysis has been done in the area of groundwater management, although the subject is by no means yet closed. A lot less work can be found in the literature on fuzzy spatial optimization. Even less exists on combined problems, like the one presented here.

Indicatively, regarding groundwater, Woldt et al (1995) considered physical parameter uncertainty in modeling and prediction. In fuzzy groundwater management a paper by Bogardi appeared as early as 1983, but most of the related work is to be found in later years. Guan and Aral (2004) considered hydraulic parameter uncertainty for the pump and treat remediation problem. They use fuzzy averages for the computation of their objective functions. The paper of Gaur et al (2015) is an indication of the continuing interest in fuzzy groundwater analysis. Fuzzy spatial optimization is only a recent subject of research, as indicated by Zhou et al (2015) and Qiu et al (2015). These two papers deal with the characteristic problem of optimal land use allocation under a variety of economical, ecological and environmental constraints, conflicting with each other in most of the cases. The present paper is part of an effort to lend fuzzy features to the above mentioned OGA, especially in the context of groundwater distribution.A simple physical aquifer model underlies the optimization process, so as to stress those aspects that concern the combination of fuzziness with optimization. The formation of the objective function is done either by defuzzification, or, more interestingly by appropriate use of α-cuts. 2 PROBLEM DESCRIPTION A problem of optimal allocation of groundwater is presented. It involves both the extraction cost and the transportation cost of the water. The problem has been presented elsewhere (Sidiropoulos and Fotakis, 2009 and 2011), but it is reviewed also here so as to demonstrate its potential fuzzy characteristics. More specifically, a hypothetical piece of land is considered, which is represented as a rectangular area. A two-dimensional orthogonal grid is formed that divides the large rectangle into small land blocks or cells. A number of water wells is placed outside the rectangular area. The water pumped from the wells is transported to the individual blocks. Each one of

Figure 1. Problem definition.

the blocks receives water from one well only and the problem is to find the optimal connections of the land blocks to the wells, so as to minimize both the pumping and the transport cost (Figure 1). A constraint can be added that does not allow the drawdowns at the well positions to exceed certain bounds.

An aquifer of infinite extent and uniform hydraulic conductivity is assumed to underlie the wells. Thus, if m is the number of wells, the drawdown at well w, $w = 1, 2, \ldots, m$ is given by

In equation (1) b is the thickness of the aquifer, k the uniform hydraulic conductivity, R the radius of influence of the well and q_w is the pumping discharge of well w.

The cost of pumping is represented by

The transport cost is given by the expression

where (x_i, y_i) are the coordinates of land block i, with $i = 1, 2, \ldots, a, b$, where a and b are the dimensions of the rectangle and i is a number given to the individual block, according to a numbering system, as explained

connection of the well w to land block k and (x w i ,

y w i) are the coordinates of the well to which the block

i is connected. The set of all these connections can

be depicted as the mosaic shown in Figure 1. The

arrangement of the figure is termed a configuration

and denoted as C. Obviously, both the pumping and

the transport cost (Equations 2 and 3) are functions of

C. A more detailed description of this functionality is

given by Sidiropoulos and Fotakis (2011). In order to

stress the dependence on C, F T is written as In Equation (1) the pumping discharges depend on the water needs of the land blocks. These are considered as given. It is assumed here that the water needs are the same for all the land blocks. Let d be the constant rate of water needs for the individual blocks.Then the pumping discharge of well w will be given by where s w is the number of blocks connected to well w. Therefore, Equation (1) can now be written as Where Equation (2) is written as where Therefore the optimization problem is formulated as: Minimize the objective function A drawdown constraint can be added: where L is a maximum permitted value for the drawdowns. 3 FUZZY VERSION OF THE PROBLEM Equations (9) and (10) constitute the crisp formulation of the problem. In the present analysis, the decision variable C, namely the configuration of the water allocation to the wells is a crisp variable. Fuzziness of the results emanates from fuzziness of the hydraulic parameter k and of the water needs measure d. Let

Figure 2. Fuzzy number K.

Figure 3. Fuzzy number D.

the hydraulic conductivity be represented by the fuzzy

number K of Figure 2. Similar representation is given

to the fuzzy water needs measure D, as in Figure 3.

It is easy to deduce that the α-cut of the fuzzy

number K is

Therefore 1/K α is given by

1

K α = [k 1α , k 2α] where

Also, D α = [d 1α , d 2α]

where

Hence, for the α-cut of the fuzzy number D K

where The last fuzzy number (11) is the coefficient in Equation (10). The α-cut of (D 2 /K) is calculated again as above according to the rules of Interval Arithmetic (Klir and Yuan, 1995). Based on the above fuzzy representations, the following fuzzy versions of the problem described by Equations (9) and (10) can be formulated: Case (a) Unconstrained problem and with pumping cost only. This is the simplest case. First the function f(C) contained in (9) is minimized resulting in f min = f (C min) Then the final fuzzy-valued objective function is Case (b) Constrained problem and with pumping cost only. The objective is to find the minimum of γf(C), where γ is any point in the interval [ddk 1α , ddk 2α] as defined by Equation (12). Let be an α-cut of the fuzzy drawdown number already encountered in Equation (10). Then the drawdown restriction can be expressed as This restriction applies to the whole α-cut and is independent of the coefficient γ appearing above. Therefore, the problem is reduced to finding the minimum of f under the constrained (14). The minimizing configuration will be valid for the same α-cut and it will be denoted as C αmin .Then the corresponding α-cut of the objective function will be Case (c) Unconstrained problem with both pumping and transport cost. The objective function is a fuzzy number for each configuration C. The centroid of F is computed for each C and it is denoted as F c (C). Then the problem is reduced to minimizing the function F c (C). The computation of the centroid is facilitated by the previous computation of α-cuts.

Case (d)

Constrained problem with both pumping and transport

cost.

Let

be the fuzzy number appearing in Equation (13) and

let G c (C) be its centroid. Then this problem can be

reduced to minimizing the objective function F c (C) of

the previous case subject to G c (C) ≤ L

Case (e)

The approach of the previous problem rests on defuzzi

fication. This methodology has also been followed by

Guan and Aral (2004) in their fuzzy groundwater opti

mization problem. Another approach based on α-cuts

eill be presented now.

Let γ be any number in an α-cut of the coefficient

fuzzy number of Equation (12), i.e.

γ ∈ [ddk 1α , ddk 2α]

Then an objective function of the following form will

result:

It can easily be shown that

Therefore,

But

Hence,

This result shows that the minimum is a monotone

increasing function of γ.Therefore, the extreme values

of F min will be attained at the end points of the interval

defined by the a-cut. On the other hand the restrictions

on drawdowns will be taken as in case (b) above, i.e.

uniform for the whole α-cut.

Hence, for each α-cut the following two problems

will have to be solved:

Minimize ddk 1α f(C) + τ(C), subject to β 2α ≤ L, as

defined in Eq. 14 and obtain C ddk1α min

Minimize ddk 2α f(C) + τ(C), subject to β 2α ≤ L, as

defined in Eq. 14 and obtain C ddk2α min Figure 4. Case (a). Figure 5. Case b. Then the minimizing function will have the form of a fuzzy number, the typical α-cut of which will be: 4 RESULTS – DISCUSSION The aquifer of infinite extent has a hydraulic conductivity of k c = 0.6 × 10 −3 m/s and a thickness of 50 m. k c = 0.2 × 10 −3 m/s in Figure 1. The water needs, uniform for all land blocks are set equal to d c = 10 m 3 /s. k c = 0.2 × 10 −3 m/s in Figure 2. The result of case (a) is shown in Figure 4, as a fuzzy number for the objective function. The central value of the diagram (17223.9) is also computed from the application of the genetic algorithm with crisp inputs. For case (b) the genetic algorithm was applied once for each α-cut level, namely for α= 0.1, 0.2, . . . , 0.9. The end points of the α-cut were then easily determined, as explained above.The resulting fuzzy number is shown in Figure 5. Case (c) is unconstrained as case (a), but due to the inclusion of the transport cost, the centroid of the function fuzzy number was used for the formation of the objective function. The resulting fuzzy number is

Figure 6. Case c.

Figure 7. cases (d) and (e). The inner circles (red color)

correspond to case (d) and the outer circles (blue color)

correspond to case (e).

shown in Figure 6. The central value (18567) is also

found from the crisp genetic algorithm. The objective

function, as an average, has a higher value (19826).

Case (d) is analogous to case (c). It yields higher values for the objective function, because of the con

straint involved. The resulting fuzzy number is of the

same form as in case (c).

For case (e) the genetic algorithm was run twice

for each α-cut level, in order to determine the two

ends of the resulting function α-cut. Figure 7 shows

the resulting function fuzzy numbers jointly. It can be

seen that fuzziness is greater in case (e).

It needs to be noted here that the horizontal axes in

Figures 4, 5, 6 and 7 represent values of the objective

functions, as these were defined above under cases

(a), (b), (c) and (d), respectively. All of these function

values appear in their raw, un-normalized forms.

5 CONCLUSION

A water extraction and allocation problem has been

presented with the purpose of enhancing with fuzzy

characteristics the spatial genetic algorithm(OGA) and

more generally to introduce fuzziness to a particular

category of problems.

The method of α-cuts was employed in order to

Fuzzy logic uses on eutrophication and water quality predictions

G.K. Ellina & I. Kagkalou

Democritus Univercity of Thrace, Department of Civil Engineering, Xanthi, Greece

ABSTRACT: The interpretation and prediction of physical, chemical and biological functions taking place in

freshwater ecosystems have been studied until today using widely available empirical and dynamic models and multi-criteria analysis methods.The purpose of this study is to investigate the eutrophication factors (temperature, NO_3^-, TP, Secchi depth and chlorophyll-a) in a highly variable system of Lake Karla (Thessaly, Greece) using Fuzzy Logic. Furthermore, we develop suitable methods for Water Quality assessment. Our goal is to decrease uncertainty towards the criteria that are used in decision making tools. More precisely, the factors being studied for the evaluation of water quality in Karla Lake are pH, Total Phosphorus and Nitrogen. In this way we can separate the Fuzzy Water Quality in classes (poor, average and good).

1 INTRODUCTION

Neural network modeling approach recently emerged as an alternative approach to simple empirical approaches for primary production estimation problems. A substantial number of research activities have been carried out on the use of neural networks in eutrophication modeling and lake management particularly during last few years (Sylaios et al. 2003, Beklioglu 2007, Sidiropoulos et al 2012).

During the past decades, the anthropogenic lake enrichment with nutrients (mainly nitrates and phosphorus) led to the rapid acceleration of the primary production rate and eutrophication level. While traditionally such development of eutrophic conditions is accustomed to nitrate and phosphorus concentrations, the shallow Mediterranean lakes, just because they are considered more complicated systems, show us a more complicated profile (Scheffer et.al. 1993). For example, factors such as: water basin geomorphology, agricultural uses, morphometry of the lake, incom ing nutrient origins, lake level variations, climate conditions and biodiversity, are directly or indirectly affecting the trophic state (Beklioglu et al.2007). As a result, trophic state estimation is considered to be a complicated and multi-criteria based topic. Literature from simple models based on experience; provide a vast number of approaches to estimate the eutrophica tion (Dillon & Rigler 1974, Rast & Lee 1978, Bartsch & Gakstatte 1978), to the point of multi-criteria sta tistical analysis and dynamic models EUTROMOD, WASP5, CAEDYEM, PCLake.

All these systems offer safer results, by providing detailed information about the spatial and temporal variation of quality parameters. Among the drawbacks mentioned about the use of dynamic models, necessity of many experimental data, great processing times and demanding calculations are the most important ones, while in the multi-criteria analysis the main disadvantage is the lowered credibility of the analysis, should the factors' impact changes. Since 1990, Fuzzy Logic is applicable, helping us manage the uncertainty of the analysis regarding environmental issues (Silvert, 2000). It is a powerful tool for exploring complex, non linear biological problems like estimating and forecasting. Fuzzy Logic models are now applied to a vast number of research studies focused into estimating the water quality and trophic state of water bodies (Kung et al 1992, Lu et al 1999). The subject of this study is to investigate the eutrophic factors found at the highly changeable and dynamic lake Karla system, with the help of Fuzzy Logic theory, thus contributing to a better understanding of the mechanisms affecting the lake's proper function. Our goal is to acquire further knowledge so we can effectively manage and protect the new ecosystem. 2 METHODS 2.1 Study area The study area is the reservoir of Karla (39 ∘ 29 ′ 02 ′′ N, 22 ∘ 51 ′ 41 ′′ E) situated in the south-eastern basin of Thessaly plain. It is about the creation of an artificial lake, with a maximum depth of 4.5 m, area of 38 km ^{2} , fed by runoff from surrounding basins and winter waters of Pinios. The climate in the eastern part of Thessaly is typical Mediterranean. The average temperature is 16–17 ∘ C and the annual average relative humidity is 67-72%. The average rainfall is 500–700 mm, while rainfall is rare from June toAugust (Sidiropoulos et al. 2012).

Figure 1. The study area lake Karla and sampling stations.

Table 1. Experimental data from the lake Karla

	Temp.	NO_3	TP	(S.D)	chl-α	
month	(∘ C)	(mg/L)	(mg/L)	(m)	(mg/m ^{3})	pH
Mar ‘12	14,4	0,2	0,05	0	76,28	9,03
Apr	17,6	0,18	0,051	0,42	145,25	8,88
May	20,6	0,43	0,05	0,37	88,03	8,94
Jun	26,7	0,49	0,186	0,34	209,57	8,82
Jul	28,7	0,31	0,079	0,32	184,1	8,47
Aug	26,6	0,37	0,079	0,2	403,58	8,7
Sep	24,1	0,54	0,038	0,19	176,68	8,65
Oct	22,8	0,1	0,05	0,33	74,87	8,3
Nov	17,5	0,1	0,05	0,33	55,91	8,6
Dec	7,9	0,16	0,05	0,485	70,07	7,94
Jan ‘13	8,1	0,2	0,05	0	37,13	7,65

Feb 8,9 0,22 0,05 0,3 73,3 8,06

Mar 11,2 0,1 0,004 0,31 43,18 7,86

Apr 21,6 0,243 0,021 0,33 118,88 7,49

May 24,7 0,189 0,042 0,4 92,1 8,49

Jun 29,7 0,32 0,021 0,37 112,1 8,7

Jul 26,6 0,426 0,022 0 46,3 8,26

2.2 Data base and model application in lake Karla

Data coming from previous field research (Chamoglou, 2014) used to refer to the average of monthly samples concerning three stations (S1, S2, S3) for the 2012 and 2013 period and referring to the following parameters: Water Temperature, Secchi Depth, (S.D.), Nitrates (NO_3^-), Total Phosphorus (TP) and Chlorophyll-a (chl-a) (Table 1). Methodology details of laboratory analysis are described by Chamoglou (2014).

2.3 Description of the fuzzy regression model

A fuzzy logic model is a non-linear description of input and output parameters. The general form of the model is as follows:

where A_i, $i = 1, 2, . . . , n$ are symmetrical fuzzy numbers. So the function involving fuzzy numbers A_i can be regarded as a probability distribution function. In this respect the fuzzy linear regression becomes a linear model in which differences between the actual values and the estimated values can be derived from uncertainty of the system. Given the data provided by the Table 1. In fuzzy linear regression the fuzzy numbers are considered: Where A

0 , A 1 , . . . , A n , i = 1, 2, . . . , n are symmetrical triangular fuzzy numbers. We determined the degree h to which we expect the data ((x 1j , x 2j , . . . x nj), y j) to be included in the referred number Y i that is μ Y i (y i) ≥ h, i = 1, 2, . . . , n. We also want the spread of fuzzy numbers Y i , i = 1, 2, . . . , m be as low as possible. Since the fuzzy numbers, A i i = 0, 1, 2, . . . , n are symmetric, are in the form: where r i is the center and c i the spread, i = 0, 1, 2, . . . , n and the function L(x) is defined: The calculation of numbers A i was performed by computing the numbers r i and c i , where i = 0, 1, . . . , n.This gives the following linear regression: 3 RESULTS – DISCUSSION 3.1 Fuzzy model In our application, we take i = 0 and i = 1, m = 17 and n = 1. Thus we get the following couples for linear regression: water temperature – chlorophyll-a, Nitrogen-chlorophyll-a, Total Phosphorus-chlorophyll-a and Secchi Depthchlorophyll-a. In Table 2 the measure of fuzziness of the input data with the output are presented. A 0 and A 1 are the fuzzy numbers calculated by the problem of linear regression (Mathematica software) for each of the parameters studied.

Table 2. Measure of fuzziness parameters with

chlorophyll-a Water temperature

Chl-a A 0 (0,0) A 1 (8.45, 6.715)

measure of 6,715

fuzziness Nitrates

Chl-a A 0 (29.015, 0) A 1 (526.45, 485.881)

measure of 485,881

fuzziness Total Phosphorus

Chl-a A 0 (-1,15, 25,11) A 1 (3036.64, 1768.7)

measure of 1768,88

fuzziness Secchi Depth

Chl-a A 0 (218.4, 181) A 1 (19.516, 0)

measure of 181

fuzziness

These fuzzy numbers are presenting all the possible

values that can be represented by our parameters in this

ecosystem.

To calculate each measure of fuzziness were

applied:

The lower the measure of fuzziness is, the more

confident we are that the parameter affects the

chlorophyll-a. It appears that the measure of fuzzi

ness is lower in the relation between temperature and

chlorophyll-a in comparison with pairs of other param

eters (Table 2). The estimation of fuzzy regression in

this relation is successful, showing low measure of

fuzziness of the chlorophyll-a. On the contrary, the

estimation of chlorophyll's concentration based on the

other parameters doesn't seem accurate because the

measure of fuzziness was bigger.

3.2 Chlorophyll-a pre-evaluation

An adaptive system was designed in order to estimate

in advance the phytoplankton biomass, as an indicator Table 3. Inputs' classification Classes Input low medium high Water temperature (∘ C) <16 11–26,2 >22 Nitrate (mg/L) <0,25 0,16–0,47 >0,37 Total Phosphorus (mg/L) <0,065 0,035–0,16 >0,12 Secchi depth (m) <0,165 0,07–0,425 >0,275 Figure 2. Classification of the studied variables in classes (Temperature, NO − 3 , TP and Secchi Depth). of trophic state (chlorophyll a) setting as inputs water temperature, nitrate and total phosphorus concentrations, and Secchi Depth. With the Matlab software (ANFIS) (Figure 2), we classified the input data so as to get a more realistic outcome. At this point the program classifies the parameters as low, medium and high (Table 3) and the more the classes are, the better approach we get. It seems that the results are equal to the data field. Indicatively, using Matlab, in June 2013 the chlorophyll-a is estimated 112 mg/m 3 while actual measurement is 112.1 mg/m 3 (Figure 3). 3.3 Estimation of Water Quality in Lake Karla At this point, fuzzy logic will be used to assess the water quality of Lake Karla developing a water quality index based on the concept of fuzzy logic. So using FIS (Matlab), we categorize the examined parameters (Figure 4). In this categorization, are formed three trapezium fuzzy numbers for each parameter with four angles, a, b, c and d, while the side “bc” is completely satisfied all the rules that have been set in this model. All possible combinations of these data sets, as model rules, have been examined by choosing this time

Figure 3. Chlorophyll-a assessment under the classes of the input data.

Figure 4. Classification of the studied variables in classes (pH, NO − 3 , TP) and the indicator Fuzzy Water Quality.

the following parameters pH, TP and NO − 3 , so as to obtain the water quality of Lake test for each possible combination of data values (Figure 5).

So, it is possible to solve this problem, for better and more accurate assessment of the water quality of Lake Karla.

By testing such different data sets it can be con cluded that the quality of the lake’s water is moderate to good (Figure 6).

This is in agreement with the assessment based on chemical monitoring under the light of the Water Framework Directive (Technical Report, River Basin Management Plan of Thessaly Greece, 2013). The status of the reservoir is further deteriorated if we con sider also the key-biological elements suggested by the Directive (Chamoglou et al., 2014) thus a more sophis ticated fuzzy model underlying species composition would be appreciated.

Since trophic status assessment of water quality is very important for the water resources management, the assessment results obtained from using only one parameter may easily mislead or bias the decision makers or managers.

Estimation of primary productivity within reser voirs of highly spatial and temporal variability is a challenging problem. The literature offers different Figure 5. Combinations of Rules. Figure 6. Water quality assessment test for April of 2013. approaches of modeling with varying degree of complexity to estimate water quality characteristics and primary productivity levels in lakes and reservoirs (Kung et al 1992 and Lu 1999). Previous investigators selected similar enclosed water bodies (lakes and reservoirs) for the application of similar methods, whereas Chl-a, was successfully applied on another Greek case study, the Vassova Lagoon (Sylaios et al. 2003) using data set originated

by field sampling and laboratory analysis. It is also confirmed that chlorophyll-a is mostly affected by water temperature than other factors.

4 CONCLUSION

Among the important problems facing the modeling process of running a water system is the high volatil ity of water quality parameters in combination with the absence of time series data. The new ecosystem of

Lake Karla is characterized by large seasonal and spa tial variability and the monitoring process coincides with the filling of the reservoir (year 2012–2013). But the need for management measures requires provision of the dynamics specific processes such as eutrophica tion. We studied the basic parameters of eutrophication using fuzzy logic methodology. The results show that the chlorophyll-a, as a basic parameter of eutroph ication, is well described by the water temperature according to the measure of fuzziness. By applying multi-criteria analysis, PCA, similar results occurred for the same ecosystem.Also the application ofANFIS (Matlab) is a useful and trustful forecasting tool for the chlorophyll-a's levels and avoid time-consuming mea surements, while minimizing errors, that can be caused by weather conditions, human factors or measuring instrument faults. Finally, utilizing the classification of the parameters pH, TP, NO − 3 referred to work by using the FIS estimate the quality of lake water based on the data we have collected; eventually we find that the water quality is moderate to good. Once the recov ery process is a complex process the results of this study could form another useful 'tool' and the next research step is to study other biotic elements using fuzzy logic.

ACKNOWLEDGEMENT

The authors would like to thank the referees for their