



Proceedings

Experimental and Numerical Analysis of 3D Dam-Break Waves in an Enclosed Domain with a Single Oriented Obstacle ⁺

Selahattin Kocaman¹, Stefania Evangelista², Giacomo Viccione^{3,*} and Hasan Güzel¹

- ¹ Department of Civil Engineering, Iskenderun Technical University, 31200 Hatay, Turkey; selahattin.kocaman@iste.edu.tr (S.K.); hasan.guzel@iste.edu.tr (H.G.)
- ² Department of Civil and Mechanical Engineering, University of Cassino and Southern Lazio, 03043 Cassino, Italy; s.evangelista@unicas.it
- ³ Department of Civil Engineering, University of Salerno, 84084 Fisciano, Italy
- * Correspondence: gviccion@unisa.it; Tel.: +39-089-96-3408
- + Presented at the 4th EWaS International Conference: Valuing the Water, Carbon, Ecological Footprints of Human Activities, Online, 24–27 June 2020.

Published: 22 August 2020

Abstract: Flood caused by a dam-breaking flow may be catastrophic for the downstream area due to the sudden discharge of large volumes of water. Besides the complex flow of the propagating dam-break wave, the presence of structures such as bridges and buildings yield free surfaces which can be accurately reproduced by means of three-dimensional Computational Fluid Dynamics (CFD) software. The prediction of the dam-break flow main features in the presence of obstacles has a crucial role in decreasing the damage. In this study, small-scale laboratory experiments were conducted to examine the problem with a single obstacle. Five ultrasonic sensors were used as measurement devices. Measurements were compared with the numerical results obtained with the FLOW-3D software, solving RANS equations with the k- ϵ turbulence closure model. A good agreement was observed.

Keywords: dam-break; FLOW-3D; experiments with fluids; ultrasonic sensors; waves

1. Introduction

Dam-break waves may have catastrophic consequences due to the sudden release of significant water volumes from the upper reservoir. They may represent, therefore, a threat for people living downstream for civil works, such as buildings, bridges, facilities, etc. [1,2]. The numerical prediction of free surface profiles, flow depths and velocities, as well as the spatial allocation of inundated areas, is of paramount importance in terms of flood hazard prevention, aiming at reducing human losses and structural damages. The presence of obstacles, e.g., buildings, bridge piers or roads, plays an important role on the water depth distribution, velocity propagation and, more in general, on the flow regime [3–5]. The way these works are oriented with respect to the main flow direction also influences the flow peculiarities. Another key aspect is related to the impact force exerted by an approaching dam-break wave [6,7].

The literature offers very few contributions regarding experimental investigations on two- and three-dimensional dam-break flows, as the measure of flow parameters under unsteady conditions is not trivial [8]. In [9], dam break investigations were carried out in a rectangular channel, comprehending dry and wet conditions for the downstream channel bottom. Cagatay and Kocaman [10] carried out experiments over a rectangular cross-section channel deriving the free surface profiles by means of digital image processing techniques. More advanced studies, involving an irregular

topography with the presence of singularities, such as contractions, have been investigated in [11– 13]. With reference to three-dimensional (3D) studies, few studies can be found, even more so when involving the presence of a building invested by a dam-break wave [14,15].

In the last two decades, dam-break flows have been mainly investigated numerically, thanks to the development of specific procedures and algorithms as well as the continuous increase of computer power. Numerical models based on two-dimensional (2D) shallow-water equations (SWEs) are commonly used, as they allow the simulation of floodplains in a reasonable amount of time often with satisfactory accuracy [16–18]. However, SWEs approaches may fail in the presence of singularities as well as steep variations in the bottom topography, as they rely on the hydrostatic distribution along the vertical direction. Therefore, models, either VOF- or FEM-based, in which the Navier–Stokes equations are averaged and solved with a turbulence model, have recently become useful tools in solving such complex rapidly varied and unsteady flow problems [19–21].

The present study aims to experimentally and numerically investigate the effects of an isolated building induced onto an approaching dam-break wave. Laboratory measurements consisting of video recordings are compared with the results obtained with the software package FLOW-3D. Results are discussed, then final conclusions are drawn.

2. The Experimental Setup

The experiments were performed on a rectangular horizontal channel, 1.00-m-long, 0.50-m-wide, and 0.35-m-high (Figure 1a). Bottom and lateral walls were made of thick glass. The upstream water domain at rest was 0.25-m-long and $h_0 = 0.15$ -m-high, mimicking the presence of a reservoir. The sluice gate, made of acrylic, 0.10-m-wide and 3-mm-thick, was placed at the center of the channel, its barycenter belonging to the streamwise plan of symmetry (dash-dot line in Figure 1a). Waterproofing was achieved by using grease oil over the vertical guides.



Figure 1. The experimental setup and removal mechanism: (**a**) plan view, (**b**) A-A cross-section (lengths expressed in cm).

In order to simulate the abrupt removal of the closing gate, a 15 kg mass connected with a steel rope was set to fall down from an elevation of 1 m, generating a fast and complete gate opening (Figure 1b). The opening time was about 0.06 s from video recordings, a period smaller than the recommended value, $1.25(h_0/g)^{1/2} = 0.15$ s, suggested by Lauber and Hager [9], with g = 9.80665 m/s² as the gravity acceleration.

Before performing an experiment, the upstream reservoir was filled with water. Dry bed conditions were realized at the downstream instead. A tall object of prismatic shape, 0.15-m-long and 0.08-m-wide, was located 0.25 m downstream from the plate. The obstacle was placed with a rotation angle of 28.0724° (Figure 1a) so that one of its diagonals overlaps the streamwise flow direction (Figure 1a). Other angles of rotations were taken into account, but corresponding results are not presented here for the sake of brevity.

In order to detect fluid movement, dye was added to the water before removing the gate. Water levels were sampled at five locations (Figure 2) using an ultrasonic sensor microsonic MIC+35/IU/TC, range 65-350 mm, at location P1 and ultrasonic sensor microsonic MIC+25/IU/TC, range 30–250 mm, at locations P2–P5. A high-speed 300 fps CCD camera was used to record the experiments' evolution. The channel was lightened by fluorescent spotlight from above to better track water interfaces. Camera was placed above as well its position matching the vertical axis of the prismatic object.



Figure 2. Allocation of the ultrasonic sensors (Plan view).

3. Numerical Model

The non-stopping increase in computer power allowed the emerging and spread of Computational Fluid Dynamics (CFD) software as predictive tools in the context of hydraulic problems. Here, the software package FLOW-3D developed by Flow Science Inc., Los Alamos, USA, was used [22] to perform simulations of unsteady flows in the experimental setup described above. Among available turbulence models, k- ε is adopted to close Reynolds-Averaged Navier–Stokes (RANS) equations [23]. Free surfaces and interfaces are solved with the Volume of Fluid (VOF) method [24] and the Fractional Area/Volume Obstacle Representation (FAVORTM).

VOF is a free-surface numerical technique for tracking and locating free surfaces and other liquid–liquid, liquid–gas interfaces. It belongs to the class of Eulerian methods, which are characterized by a computing mesh either stationary or integral to the moving domain to accommodate the evolving shape of the interfaces. The FAVORTM formulation [25] is a collection of procedures implemented in Flow-3D[®] to prevent numerical flow losses. It avoids the "zigzag" approximation of solid boundaries which may occur when the geometry is represented by a Cartesian grid, as in FLOW-3D[®]. FAVORTM package eliminates zigzag direction changes by smoothly blocking out fractional portions of grid cell faces and volumes. In addition, it is used to compute interfacial areas as well as advecting term along them, wall stresses and to enhance numerical stability.

The computational domain was discretized in a rectangular mesh. The size of grid cells was fixed to $\Delta x = \Delta y = \Delta z = 0.05$ m, resulting in 200, 100 and 36 cells along the *x*, *y* and *z* directions, respectively. The total number of cells was therefore 720,000. The channel and obstacle surfaces were assumed smooth. Simulations were run for *t* = 15 s. The time step Δt was determined according to the Courant-Friedrichs-Lewy condition. The run-time was approximately 1 h 50 min on average, on a computer with Intel Core i7 2.8 GHz 16 GB RAM.

4. Results

The present study aims to observe and provide insight for the flow behavior when the dambreak flow encounters an oriented obstacle on its route in a 3D enclosed domain. Towards this aim, new experimental data were obtained using non-intrusive measurement techniques with ultrasonic sensors due to lack of real-case dam-break data and a limited number of experimental data. In addition, the comparison of the obtained results of time-dependent water levels with the results of an RANS-based numerical model, revealed the solution ability of the numerical model to efficiently simulate a complex, rapidly varied unsteady flow caused by a dam break. After the sudden removal of the sluice gate, a dam-break wave propagates downwards (Figure 3a, t = 0.1 s). As the wave reaches the object, it interacts locally by first increasing the water depth, and the flow separates (Figure 3b, t = 0.4 s), then decreasing it as the wave propagation takes place. The flow direction is significantly affected by presence of the object. At the initial and following stages of the dam-break flow, a wake zone occurs behind the object. Re-circulation zones can also be observed between the object and the lateral walls. The surrounded flow over the object reattaches downstream with the formation of negative waves (Figure 3c, t = 1.0 s). Object affects the flow conveyance and flow depths increase due to the decrease in the propagation speed of the flood wave. Reflections of the flow from the object, sidewalls and end wall result in oblique hydraulic jumps and lateral jumps, and these increase flow complexity.



Figure 3. Time instants of video recordings. (a) t = 0.1 s, (b) t = 0.4 s, (c) t = 1.0 s.

The observed phenomenon is quite complex, involving multiple wave interactions, which can be tackled, however, with 3D numerical methods. In the present study, an RANS scheme was run in FLOW-3D to reproduce the unsteady flow pattern. Acquired water level changes by means of five ultrasonic sensors 10 s after the gate removal are given in Figure 4. As shown in Figure 2, the P1 sensor was located in the reservoir to follow its emptying, and the others (P2-P5) were placed around the obstacle. The comparison of the results showed good agreement overall. For P1, the water level progressively decreased as the reservoir empties until t = 6 s and remained nearly horizontal after t =6 s due to the limited reservoir capacity. Small oscillations occurred on the water surface in the reservoir, due to the reach and reflection of the negative wave from the channel upstream end and sidewalls. It is also noticeable that the numerical water level results were somewhat below the experimental level results during the descent in the reservoir. The water level increased rapidly with the arrival of the flood wave at the P3 sensor located upstream of the object. Then, the water level started to decrease slowly due to the decrease in water in the reservoir until t = 2 s. The water level increased again as soon as the wave reflected from the upstream end of the canal arrives. With the arrival of the reflected wave at other measurement points (P2–P5) downstream of the gate, increasing water level, and undulations on the water surface were noticeable.

In the numerical simulations, the undulations occurring on the water surface are slightly fewer compared to the experimental study. In addition, considering the time taken to reach the measurement points of the incoming wave and the time that the reflected wave passes through these points again, it can be said that the numerical method determines the propagation speed of the wave front well. The main difference between experimental and numerical results was observed immediately after the arrival of the reflected wave. This negative wave also can be considered as a moving hydraulic jump and the flow is truly three-dimensional with air entrainment. The reason for this difference may be that air interference is not taken into account in the numerical model. As a result, FLOW-3D could simulate with reasonable accuracy the behavior of the flow at the initial and further stages of a dam-break event in the presence of an obstacle.

A similar flow behavior was generally observed when the results are compared with the results of [14]. However, the study of [14] was carried out in a large-scale prismatic channel with a trapezoidal cross-section with an open downstream end. Resistive gauges were used in that study, which are intrusive and can alter the flow behavior. The present study was conducted, instead, in a small-scale rectangular channel in an enclosed domain and used ultrasonic sensors enabling the non-intrusive measurement of the flow. In this study, a more complicated small-scale current

environment was created, with the closed end wall, and good data were obtained to validate numerical studies.

In the study by Soares-Frazão and Zech [14], the water depth increased significantly in the close region upstream to the obstacle due to the reflection of the flood wave from the longer face (in planview) of the rectangular obstacle facing the dam. In the current study, since the obstacle has orientation with the 90-degree sharp corner facing the incoming dam-break wave in plain view, i.e., one of its diagonals overlapping the streamwise flow direction, the reflection of the flow and rise in water level were more limited instead, after the wave impacts the obstacle. When the obstacle is placed at a right angle in relation to the main flow direction, the water level swelling in front of the obstacle is more pronounced.

It is noticeable that (non-intrusive) ultrasonic sensor results have less signal interference compared to (intrusive) wave resistive gage results in [14]. Comparing the two measurement methods will be useful in future studies.



Figure 4. Water depth comparison at the locations P1–P5. (a) P1, (b) P2, (c) P3, (d) P4, (e) P5.

5. Conclusions

The presence of a single object of prismatic shape, interacting with an upcoming dam-break wave in a rectangular channel, was assessed. Laboratory experiments were carried out, placing the object at the center of the channel. When the flood wave hits the obstacle, the route of the flood wave is changed, the depth of the water increases and a complex flow medium occurs that includes hydraulic jumps moving in different directions due to the obstacle and side walls. Flow pattern was inherently three-dimensional with a complex shape of the free surface near the object. The acquired experimental measurements of water levels taken with five ultrasonic sensors were compared with an RANS-based FLOW-3D numerical simulation software, with a k- ϵ turbulence model for closure. A good agreement between experimental and computed results was found. The results indicated that FLOW-3D can be a useful tool in simulating dam-break kinds of flow phenomena.

The ultrasonic sensors have also been shown to be used effectively in measuring water levels for rapidly varied unsteady flow such as dam-break.

Due to the limitations of computing power for the detailed solution of large-scale dam-break problems using RANS-based CFD models or particle-based numerical models, like Smoothed Particle Hydrodynamics (SPH) and Lattice Boltzmann (LB), these new experimental data can be used as a case study for model validation.

The obstacle located at different orientation angles with respect to the main flow direction and different tailwater levels and the combination of multiple obstacle configurations can be considered in a future study.

Author Contributions: S.K., S.E., G.V. and H.G. conceived and designed the experiments; S.K. and H.G. performed the experiments; S.K., S.E., G.V. and H.G. analyzed the data and drawn the conclusions; S.K., S.E., G.V. and H.G. wrote and revised the paper. All authors have read and agree to the published version of the manuscript.

Funding: This research received no external funding.

Conflicts of Interest: The authors declare no conflict of interest.

References

- Norin, S.V.; Belikov, V.V.; Aleksyuk, A.I. Simulating flood waves in residential areas. *Power Technol. Eng.* 2017, 51, 52–57, doi:10.1007/s10749-017-0782-3.
- 2. Hooshyaripor, F.; Tahershamsi, A. Effect of reservoir side slopes on dam-break flood waves. *Eng. Appl. Comput. Fluid* **2015**, *9*, 458–468, doi:10.1080/19942060.2015.1039630.
- 3. Evangelista, S.; Altinakar, M.; Di Cristo, C.; Leopardi, A. Simulation of dam-break waves on movable beds using a multi-stage centered scheme. *Int. J. Sediment Res.* **2013**, *28*, 269–284, doi:10.1016/S1001-6279(13)60039-6.
- De Marinis, G.; Di Cristo, C.; Evangelista, S.; Leopardi, A. Dam Break Waves on stepped bottom. In Proceedings of the XXXIII IAHR Int. Biennal Congress 2009, Vancouver, BC, Canada, 8–14 August 2009; pp. 1262–1268.
- Evangelista, S.; Greco, M.; Iervolino, M.; Leopardi, A.; Vacca, A. A new algorithm for bank-failure mechanisms in 2D morphodynamic models with unstructured grids. *Int. J. Sediment Res.* 2015, 30, 382–391, doi:10.1016/j.ijsrc.2014.11.003.
- 6. Evangelista, S. Experiments and numerical simulations of dike erosion due to a wave impact. *Water* **2015**, *7*, 5831–5848, doi:10.3390/w7105831.
- Viccione, G.; Ferlisi, S.; Marra, E. A numerical investigation of the interaction between debris flows and defense barriers. In *Advances in Environmental and Geological Science and Engineering, Proceedings of the 8th International Conference on Environmental and Geological Science and Engineering (EG'15), Salerno, Italy, 27–29 June 2015;* WSEAS Press: Salerno, Italy, 2015; pp. 332–342, ISBN 978-1-61804-314-6.
- Liu, X.; Wang, B.; Zhang, J.; Liu, W.; Song, J. A Non-intrusive measurement of full-field water depth and its application in experimental investigations of dam-break flows. *Gongcheng Kexue Yu Jishu/Adv. Eng. Sci.* 2019, *51*, 52–58, doi:10.15961/j.jsuese.201800689.
- 9. Lauber, G.; Hager, W.H. Experiments to dam-break wave: Horizontal channel. J. Hydraul. Res. 1998, 36, 291–308, doi:10.1080/00221689809498620.
- Cagatay, H.; Kocaman, S. Experimental study of tail water level effects on dam-break flood wave propagation. In Proceedings of the River Flow 2008 International Conference on Fluvial Hydraulics, Cesme, Izmir, Turkey, 3–5 September 2008; Volume 1, pp. 635–644.
- 11. Soares-Frazao, S. Experiments of dam-break wave over a triangular bottom sill. *J. Hydraul. Res.* **2007**, *45*, 19–26, doi:10.1080/00221686.2007.9521829.
- 12. Aureli, F.; Mignosa, P.; Tomirotti, M Dam-break flows in presence of abrupt bottom variations. In Proceedings of the XXVIII IAHR Cong 1999, Graz, Australia, 12–15 December 1999; pp. 163–171.
- 13. Bellos, V.; Soulis, J.V.; Sakkas, J.G. Experimental investigation of two-dimensional dam-break induced flows. *J. Hydraul. Res.* **1992**, *30*, 47–63, doi:10.1080/00221689209498946.
- 14. Soares-Frazão, S.; Zech, Y. Experimental study of dam-break flow against an isolated obstacle. *J. Hydraul. Res.* **2007**, *45*, 27–36, doi:10.1080/00221686.2007.9521830.

- 15. Aureli, F.; Maranzoni, A.; Mignosa, P.; Ziveri, C. Dam-break flows: Acquisition of experimental data through an imaging technique and 2D numerical modelling. *ASCE J. Hydraul. Eng.* **2008**, *134*, 1089–1101, doi:10.1061/(ASCE)0733-9429(2008)134:8(1089).
- 16. Brufau, P.; Garcia-Navarro, P. Two-dimensional dam-break flow simulation. *Int. J. Numer. Methods Fluids* **2000**, *33*, 35–57, doi:10.1002/(SICI)1097-0363(20000515)33:1<35::AID-FLD999>3.0.CO;2-D.
- 17. Liang, Q.; Marche, F. Numerical resolution of well-balanced shallow water equations with complex source terms. *Adv. Water Resour.* **2009**, *32*, 873–884, doi:10.1016/j.advwatres.2009.02.010.
- 18. Quecedo, M.; Pastor, M.; Herreros, M.I.; Merodo, J.A.F.; Zhang, Q. Comparison of two mathematical models for solving the dam break problem using the FEM method. *Comput. Methods Appl. Mech. Eng.* **2005**, *194*, 3984–4005, doi:10.1016/j.cma.2004.09.011.
- 19. Shigematsu, T.; Liu, P.L.F.; Oda, K. Numerical modelling of the initial stages of dam-break waves. *J. Hydraul. Res.* **2004**, *42*, 183–195, doi:10.1080/00221686.2004.9728381.
- 20. Biscarini, C.; Francesco, D.S.; Manciola, P. CFD modelling approach for dam break flow studies. *Hydrol. Earth Syst. Sci.* **2010**, *14*, 705–718.
- 21. Ozmen-Cagatay, H.; Kocaman, S. Dam-break flows during initial stage using SWE and RANS approaches. *J. Hydraul. Res.* **2010**, *48*, 603–611.
- 22. Flow Sciences Inc. FLOW-3D User Manual; Flow Sciences Inc.: Santa Fe, NM, USA, 2016.
- 23. Launder, B.E.; Spalding, D.B. The numerical computation of turbulent flows. *Comput. Methods Appl. Mech. Eng.* **1974**, *3*, 269–289, doi:10.1016/0045-7825(74)90029-2.
- 24. Mampaey, F.; Xu, Z.A. Simulation and experimental validation of mould filling. In Proceedings of the Modeling of Casting, Welding and Advanced Solidification Processes VII, London, UK, 10–12 September 1995.
- Hirt, C.W.; Sicilian, J.M. A porosity technique for the definition of obstacles in rectangular cell mesh. In Proceedings of the Fourth International Conference on Ship Hydrodynamics, Washington, DC, USA, 24– 27 September 1985.



© 2020 by the authors. Licensee MDPI, Basel, Switzerland. This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution (CC BY) license (http://creativecommons.org/licenses/by/4.0/).