22nd Canadian Hydrotechnical Conference
22e Conférence canadienne d'hydrotechnique

Water for Sustainable Development : Coping with Climate and Environmental Changes
L'eau pour le développement durable: adaptation aux changements du climat et de l'environnement
Montreal, Quebec, April 29 – May 2, 2015 / Montréal, Québec, 29 avril – 2 mai 2015

NUMERICAL SIMULATION OF DAM-BREAK FLOWS USING DEPTH-AVERAGED HYDRODYNAMIC AND THREE-DIMENSIONAL CFD MODELS

D.M. Robb[1] and J.A. Vasquez[1]

1. Northwest Hydraulic Consultants Ltd, North Vancouver, BC, Canada

ABSTRACT: This paper investigates the capability of three different numerical models to simulate sudden dam-break flows in the presence of an obstacle. The numerical results from a two-dimensional (2D) hydrodynamic depth-averaged model and two fully three-dimensional (3D) computational fluid dynamics (CFD) models are presented. The models selected for this study are: (1) the free and open-source code TELEMAC-2D, (2) the commercially-available CFD software package FLOW-3D, and (3) the free and open-source CFD code OpenFOAM. TELEMAC-2D is based on the solution of the shallow-water (Saint-Venant) equations, which neglect vertical velocities and accelerations. Both FLOW-3D and OpenFOAM are based on the solution of the Navier-Stokes equations along with the volume of fluid (VOF) method to track the location of the free surface at the air-water interface. The numerical results are compared to existing experimental data from two flume experiments conducted as part of the European IMPACT project. Flow conditions in both experiments are complex due to the presence of hydraulic jumps, reflecting waves, and wet-dry fronts. All three numerical models agree well with the water levels recorded in the experiments, especially for the triangular bottom sill case. The results from the depth-averaged model compare well with the experimental data and with the numerical results from the more sophisticated and computationally demanding 3D CFD models. The results from the CFD models show little improvement over those from the depth-averaged model, with the exception of one location in the vicinity of a moving hydraulic jump.

Keywords: dam break; volume of fluid; shallow-water equations; CFD

1. INTRODUCTION

Dam failures have been responsible for devastating consequences including loss of life, destruction of property, damage to infrastructure, and significant ecological impacts. One of the most common concerns with dam failures is extreme flooding in the downstream reach. Unlike flooding from rainfall or snowmelt, which can often be predicted days or weeks in advance, dam-break flooding is abrupt, leaving little time for evacuation of residents or protection of property. Consequently, accurate estimates of flood arrival time, flow depths, and velocities are important for flood hazard assessment and emergency planning. However, predicting these quantities is challenging and requires the use of suitable mathematical and numerical models.

A dam-break wave results from a sudden release of a mass of fluid in a channel. Upon release, a surge front develops, characterized by rapid variations in flow depth and velocity. The surge front is a sudden discontinuity, generating a shock condition in the fluid flow. These flow properties present a challenge for many mathematical and numerical models. Theoretical studies for idealized dam-break waves date back as far as Ritter (1892), who derived an analytical solution for an instantaneous dam-break flow in a horizontal frictionless channel. Further developments include the effect of bed resistance (Dressler 1952), finite length reservoirs (Hunt 1984), and initially dry, sloping channels with turbulent motion (Chanson 2009). These solutions provide a way of predicting the flow characteristics of a dam-break wave in a few idealized cases. However, for more complicated cases (e.g., involving obstacles or irregular topography), these analytical solutions have limited applicability. As a result, numerical models are much-needed tools for evaluating the hydraulic quantities important for flood hazard assessment and emergency planning of dam failures.

Experimental studies of dam-break flows have primarily focused on water level measurements (Frazão and Zech 2002; Miller and Hanif Chaudhry 1989). Fewer studies have included measurements of the velocity field (Fraccarollo and Toro 1995; Soares-Frazão and Zech 2007) and dynamic pressure loads from the impact of a dam-break wave on a structure (Aureli et al. 2015; Lobovský et al. 2014). Many of these experimental studies have yielded high-quality datasets that have been subsequently used to validate numerical models. Of particular interest to this paper are two flume experiments conducted as part of the European IMPACT project (*J. Hydraulic Res.*, 45(sup1)). The first experiment, conducted by Soares-Frazão (2007), involves the sudden dam break over an initially dry bed, where a wave climbs up and over a triangular bottom sill obstacle. The measurements from this experiment are used to validate numerical models aimed at simulating flood propagation over dry beds with irregular topography. In the second experiment, designed by Soares-Frazão and Zech (2007), a dam-break wave impacts an isolated obstacle oriented skewed to the main flow direction. The dataset from this experiment is commonly used to validate numerical models for flood propagation in the presence of obstacles.

Two-dimensional (2D) depth-averaged modelling is widely used to evaluate dam-break flows. Various approaches have been developed to accurately simulate the wetting and drying processes at the wave front, impact against obstacles or buildings (Erpicum et al. 2009), and real bathymetry (Hervouet and Petitjean 1999). These models are commonly validated by comparing the computed flow depths and velocities to analytical solutions for idealized cases or to measurements in a laboratory. Several recent 2D model studies have used the data provided by Soares-Frazão (2007), and Soares-Frazão and Zech (2007) to validate their numerical models (Hou et al. 2014; Kim et al. 2010; Murillo et al. 2009; Vasquez and Roncal 2009; Yu and Duan 2012). 2D depth-averaged models are based on the solution to the shallow-water equations, which assume that the vertical velocities are significantly smaller than the horizontal ones. As a result, the pressure field is found to be hydrostatic. This assumption prevents shallow-water models from capturing some observed hydraulic features in dam-break flows, especially in the near-field and during the initial stages of a dam break. During a brief period after a dam fails (or a gate is opened in the idealized case), the flow is primarily influenced by vertical acceleration due to gravity and the shallow-water assumptions are violated (Biscarini et al. 2010). Also, when a dam-break wave impacts an obstacle, there is strong curvature of the free surface and non-hydrostatic vertical accelerations develop in the near-field, close to the obstacle walls (Aureli et al. 2015).

Recent studies have applied fully three-dimensional (3D) models to dam-break flows, in efforts to overcome some of the shortcomings of shallow-water models. Many studies focused on particular aspects of dam-break flows, where the shallow-water assumptions break down, for example, the initial stages of a dam break (Oertel and Bung 2012; Ozmen-Cagatay and Kocaman 2010; Shigematsu et al. 2004), near-field and turbulent dam-break flow behaviour (LaRocque et al. 2013), and estimating dynamic loads due to wave impact on an obstacle (Aureli et al. 2015). The 3D models are traditionally based on the solution to the Navier-Stokes equations along with the volume of fluid (VOF) or level set method to track the free surface. Aureli et al. (2015) also used the Lagrangian method, smoothed particle hydrodynamics (SPH), to estimate the impact load on an obstacle in a dam-break flow. Several 3D model studies have used the data provided by Soares-Frazão (2007) and Soares-Frazão and Zech (2007) to validate their numerical models (Biscarini et al. 2010; Vasquez and Roncal 2009). While there are certain hydraulic features in dam-break flows that cannot be captured using shallow-water models, the application of 3D models for field-scale simulations is computationally expensive, and depending on the desired outcomes, may not yield more accurate information than the shallow-water model.

This paper compares the capability of 2D shallow-water, and fully 3D computational fluid dynamics (CFD) models to predict the hydraulic conditions caused by a dam-break wave in the presence of an obstacle. The 2D model is TELEMAC-2D, a free and open-source code that solves the shallow-water equations. Results from two 3D models are presented: (1) the commercial software package, FLOW-3D, and (2) the free and open-source code, OpenFOAM. Both 3D models simulate two-phase flow using the VOF method to track the free surface. The main advantages and shortcomings of the three models are presented. The objective of this paper is to compare three models commonly used by engineers to simulate open channel flows and further verify their applicability for dam-break flow simulations. The rest of the paper is organized as follows: the three numerical models used in this study are described in Section 2, the numerical results are compared to existing experimental data in Section 3, and conclusions are drawn in Section 4.

2. NUMERICAL MODELS

The dam-break experiments were simulated using a 2D shallow-water model, TELEMAC-2D, and two 3D CFD VOF models, FLOW-3D and OpenFOAM. The main features of each model are presented in this section.

2.1. TELEMAC-2D

TELEMAC-2D is a free and open-source hydrodynamic code, commonly used in both research and industry. This model uses an unstructured triangular discretization to solve the shallow-water equations, which neglect vertical velocities and assume a hydrostatic pressure distribution. The adopted model is a finite volume scheme using the Harten-Lax-van Leer-contact (HLLC) approximate Riemann solver, described further in Toro (2009). This numerical approach was chosen because it is known to capture fluid flow involving shock waves, surges, hydraulic jumps, and bore interactions without producing spurious oscillations or excessive numerical diffusion (Fraccarollo and Toro 1995; Yu and Duan 2012). Turbulent effects were not considered for this model.

2.2. FLOW-3D

FLOW-3D is commercially-available CFD software package commonly used to model hydraulic structures such as spillways, stilling basins, and water intakes. It has also been successfully applied to model dam-break flows (Vasquez and Roncal 2009; Ozmen-Cagatay and Kocaman 2010; Oertel and Bung 2012). FLOW-3D solves the Reynolds-averaged Navier-Stokes (RANS) equations and uses the VOF method to track the free surface (Hirt and Nichols 1981). The grid is not boundary-fitted; instead, an immersed boundary method is used, where solid geometries embedded within the grid are represented using the Fractional Area/Volume Obstacle Representation (FAVOR) (Hirt and Sicilian 1985). The numerical solver can apply different turbulence models and the RNG k-ε turbulence model was selected for this study.

2.3. OpenFOAM

OpenFOAM is a free and open-source suite of C++ libraries designed for the development of numerical solvers for continuum mechanics problems, including CFD applications. The dam-break experiments were simulated using the OpenFOAM solver, InterFoam. This solver models an incompressible and immiscible two-phase (water-air) system, using the VOF method to track the free surface on the air-water interface. The model uses the finite volume method to solve the RANS equations. The pressure-implicit with splitting operators (PISO) scheme is used for pressure-velocity coupling (Issa 1986). To be consistent with the FLOW-3D simulations, we used the RNG k-ε turbulence model for the OpenFOAM simulations presented herein.

3. NUMERICAL RESULTS

The models were tested using existing experimental data from two flume experiments simulating dam-break flows. The following section provides a description of each test and the associated numerical results.

3.1. Dam-break wave over a triangular bottom sill

As a first test case, the propagation of a dam-break wave over a triangular bottom sill was simulated. The experiment was designed by Soares-Frazão (2007). The purpose of selecting this test case was to compare the ability of the models to simulate flood propagation over a dry bed with irregular topography.

3.1.1 Experimental setup

The experimental setup is sketched in Figure 1. Experiments were conducted in a straight rectangular channel, 5.6 m long and 0.5 m wide, closed at both ends by vertical walls. The upstream reservoir is 2.39 m long and is initially filled with 0.111 m of water at rest, held in place by a sliding gate that acts as a dam. The channel is initially dry between the gate and the triangular bump. The symmetric bump is 0.065 m high and slopes at 0.14 on both sides. Downstream of the bump, a pool contains 0.02 m of water at rest. The Manning friction coefficient is 0.011 s/m$^{1/3}$. Three resistive gauges were used to record water levels at the locations indicated in Figure 1. Further details of the experiment are provided in Soares-Frazão (2007).

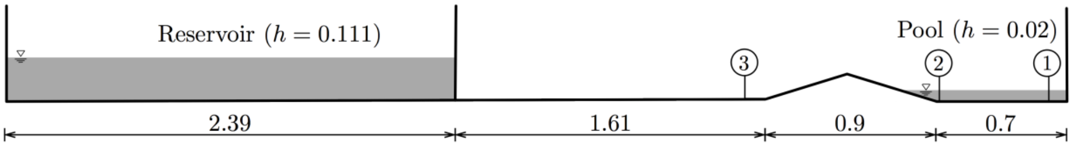

Figure 1. Experimental setup and initial conditions for *Dam-break wave over a triangular bottom sill*, all units in metres, water level gauges labelled 1 to 3, side view.

3.1.2 Physical description

Following the sudden opening of the gate, the flooding wave from the reservoir spreads over the dry bed toward the bump. As the wave reaches the bump, part of the wave is reflected and forms a bore propagating in the upstream direction, while the remaining downstream-moving wave continues over the dry bump. After the wave passes over the top of the bump, it flows on the dry downward sloping side of the bump until it reaches the pool of water at rest. Once the wave reaches the pool, the wave front is slowed abruptly and forms a bore that continues in the downstream direction. The bore reflects against the downstream end of the channel and travels toward the bump. During the experiment, several reflections of the flow occur both at the bump and the channel ends. The observed phenomenon was simulated by using the models presented in Section 2.

3.1.3 Model setup

For the TELEMAC-2D model, the computational domain was discretized into an unstructured triangular mesh. The simulations were conducted with mesh resolutions of 0.1 m, 0.067 m, 0.05 m, 0.02 m, and 0.01 m, with mesh independence occurring at 0.02 m. The results presented herein are from computations using the 0.02 m mesh having approximately 7,000 nodes and 13,000 elements. A Manning's roughness coefficient of 0.011 s/m$^{1/3}$ was adopted for the channel bottom.

For both the FLOW-3D and OpenFOAM models, the computational domain was discretized into hexahedral cells 0.1 m long, 0.1 m wide, and 0.005 m high. The resulting meshes consisted of 896,000 cells. All boundaries were assumed to be rigid walls except for the top boundary, where a pressure outlet at constant atmospheric pressure was imposed. Simulations were also conducted with a finer mesh consisting of cells 0.05 m long, 0.05 m wide and 0.005 m high. The results with the finer mesh were virtually the same as with the coarser mesh, which suggests that the solution is mesh independent at the size of the coarser mesh reported herein.

3.1.4 Model results

Figure 2 shows the measured and computed water level time history at the three gauge locations. The numerical results show very good agreement with the experimental data at all three gauges. The model predicts the arrival time of the wave front over the dry bed very well. The surge formation on the downstream side of the bump and subsequent wave reflections are accurately modelled. Overall, the models predict both the magnitude and the timing of the measured water levels with a high degree of accuracy. Table 1 shows the root-mean-square (RMS) error between the measured and computed water levels for each model. The modelled water levels are generally within 3 to 6 mm of the measured values. Considering there is experimental error associated with repeatability and instrumentation, the modelled results are quite accurate. This test confirms that all three models can accurately simulate dam-break waves over a dry bed with irregular topography. While this free-surface flow has many complex elements including wet-dry fronts, surges, reflecting waves, and jet-like flow characteristics, the error analysis shows that there is little advantage of using a 3D model for predicting water levels. Furthermore, as demonstrated by the TELEMAC-2D results, turbulence modelling is not required for predicting the water levels in this experiment.

Table 1. Error Analysis: *Dam-break wave over a triangular bottom sill*.

Gauge	**RMS Error (cm)**		
	Telemac-2D	**FLOW-3D**	**OpenFOAM**
1	0.4	0.5	0.3
2	0.6	0.5	0.4
3	0.4	0.4	0.4
Average	0.5	0.5	0.4

3.2. Dam-break flow against an isolated obstacle

The experiment was designed by Soares-Frazão and Zech (2007). The measurements taken during the experiment make up a dataset that can be used for numerical model validation for problems involving fast transient flow and complex topographies. The purpose of selecting this test case was to compare the ability of the numerical models to simulate dam-break flows in the presence of an obstacle (i.e., a fast transient flow problem with complex topography).

3.2.1 Experimental setup

Experiments were conducted in a straight open channel 35.8 m long and 3.6 m wide, as shown in Figure 3. The channel is rectangular in cross-section except near the bed, where it is cut to form a trapezoidal shape. An impermeable wall with a 1 m wide sliding gate is placed 6.9 m from the flume entrance. Initially, 0.4 m of water is filled upstream of the gate, representing a reservoir. Downstream of the gate, an isolated obstacle, representing a building, is oriented 64° to the side of the flume. A thin layer of 0.02 m of water is initially placed in the channel downstream of the gate. The Manning friction coefficient is 0.010 $s/m^{1/3}$. Six resistive gauges were used to record water levels at the locations indicated in Figure 3. Data from the five gauges near the obstacle were used to validate the numerical results. Further details of the experiment are provided in Soares-Frazão and Zech (2007).

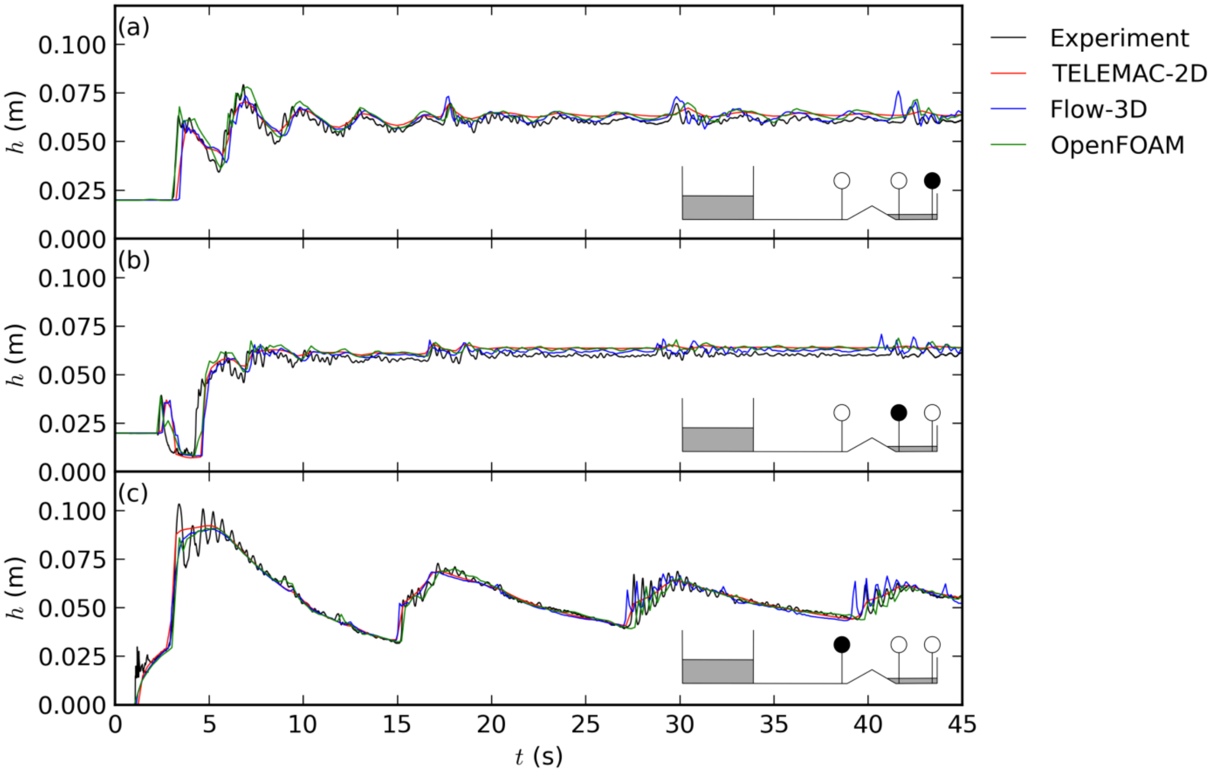

Figure 2. Comparison between experimental and numerical results for *Dam-break wave over a triangular bottom sill*: water level time series at (a) gauge 1, (b) gauge 2, and (c) gauge 3.

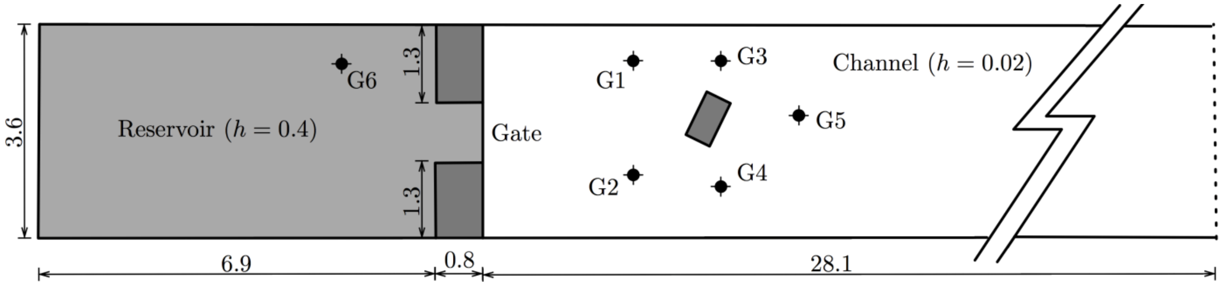

Figure 3. Experimental setup and initial conditions for *Dam-break flow against an isolated obstacle*, all units in metres, water level gauges labelled G1 to G6, plan view.

3.2.2 Physical description

After the sudden opening of the gate, a dam-break wave flows radially out of the reservoir and impacts the side walls of the flume and the obstacle. Upon impact, the flow transitions from supercritical to subcritical, forming moving hydraulic jumps along the side walls and upstream of the obstacle. As the flow splits on either side of the obstacle, it interacts with waves reflected from the side walls. This interaction forms a series of shock waves (hydraulic jumps) that cross each other. As the reservoir empties into the downstream channel, the water levels in the channel rise while inflow discharges decrease, causing transient conditions with moving hydraulic jumps. The jump upstream of the obstacle is initially oblique, resulting in water levels on the left side of the channel (gauge G1) to be initially higher than on the right side (gauge G2).

3.2.3 Model setup

For the TELEMAC-2D model, simulations were conducted with mesh resolutions of 0.1 m, 0.067 m, 0.05 m, and 0.02 m, with mesh independence occurring at 0.05 m. The results presented herein are from computations using the 0.05 m mesh, having approximately 33,000 nodes and 66,000 elements. A Manning's roughness coefficient of 0.010 s/m$^{1/3}$ was adopted for the channel bottom. All boundaries were treated as rigid walls except the downstream end of the channel, where a constant water level of 0.02 m was imposed.

A variable resolution, boundary-fitted mesh was generated for the OpenFOAM model. The boundary-fitted mesh consisted of predominantly hexahedral cells with maximum dimensions of 0.03 m long, 0.03 m wide, and 0.01 m high, refining near the isolated obstacle to minimum dimensions of 0.02 m long, 0.02 m wide and 0.01 m high. The mesh had approximately 3,800,000 cells in total. Because FLOW-3D uses an orthogonal grid, it is not possible to use identical meshes for the 3D models. The FLOW-3D grid consisted of hexahedral cells 0.035 m long, 0.02 m wide and 0.01 m high, having approximately 4,200,000 cells in total. No-slip boundary conditions were imposed on the channel bed and the vertical walls. A roughness height of 0.0002 m was adopted for the channel bottom. On the top boundary, a pressure outlet at constant atmospheric pressure was imposed. An open boundary condition was adopted for the downstream end of the channel, allowing the flow to pass freely out of the computational domain.

3.2.4 Model results

Figure 4 shows a time sequence comparing the 2D and 3D numerical results during the first ten seconds of the experiment. The results shown in Figure 4 were computed using TELEMAC-2D and OpenFOAM. The 2D results capture the wave propagation and the location of the hydraulic jumps very well. The shallow-water assumption and the absence of a turbulence model prevent the model from accurately describing regions of fully three-dimensional turbulent flow, for example, in the surface roller of a hydraulic jump. Instead, the 2D model shows smooth, abrupt, nearly vertical transitions between the supercritical and subcritical flow regimes. The 3D results provide a more accurate description of the flow regime in the areas where three-dimensional and turbulent effects are important.

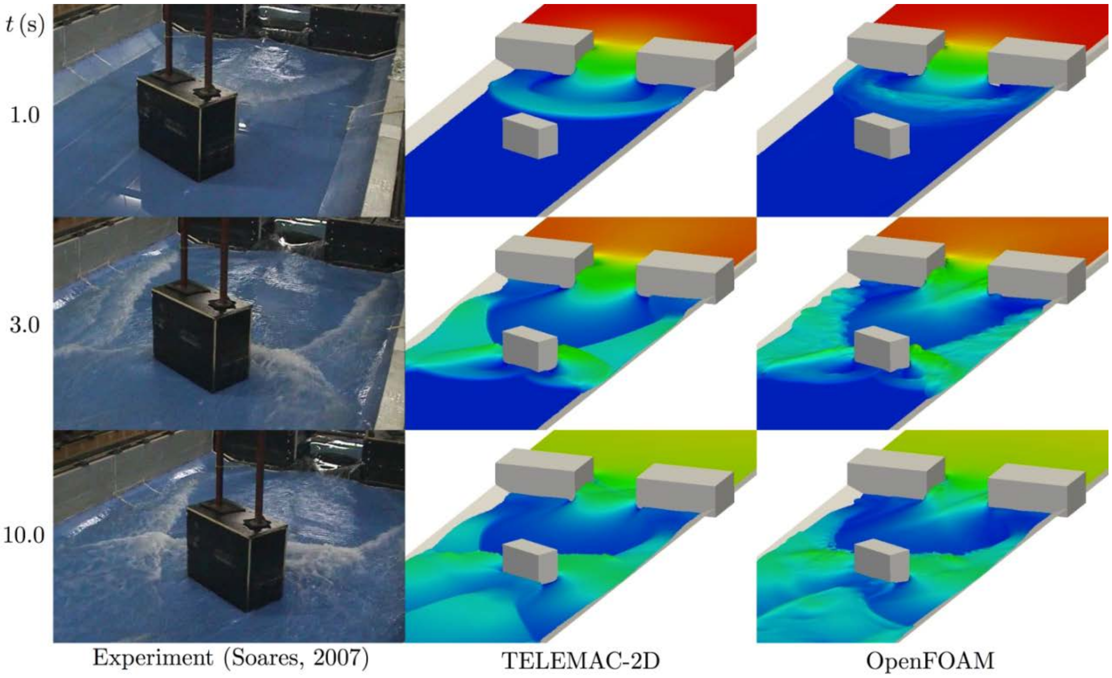

Figure 4. Time sequence of computed results at times 1 s, 3 s, and 10 s. The colour gradient represents the free-surface elevation from 0 m (blue) to 0.4 m (red).

Figure 5 shows the measured and computed water level time history at the five gauge locations near the obstacle. The numerical results show very good agreement with the experimental data at all five gauges. Overall, the models predict both the magnitude and the timing of the measured water levels reasonably well, although results at gauge G2 warrant further discussion. Experimental data at gauge G2 show a gradual increase in water level between 13 s and 20 s, as a hydraulic jump migrates upstream. Several academic (Soares-Frazão et al. 2003) and commercial (Neelz and Pender 2013) 2D models have been used to simulate this experiment, and at gauge G2, most have either missed the jump completely or predicted an abrupt increase in water level, similar to the present results obtained by TELEMAC-2D (Figure 5). Although FLOW-3D is a fully three-dimensional model with a turbulence treatment, it also predicts a relatively abrupt increase in water level at gauge G2. In this study, OpenFOAM has most accurately predicted the gradual upstream migration of the jump recorded at gauge G2, producing the lowest RMS error for this gauge (Table 2).

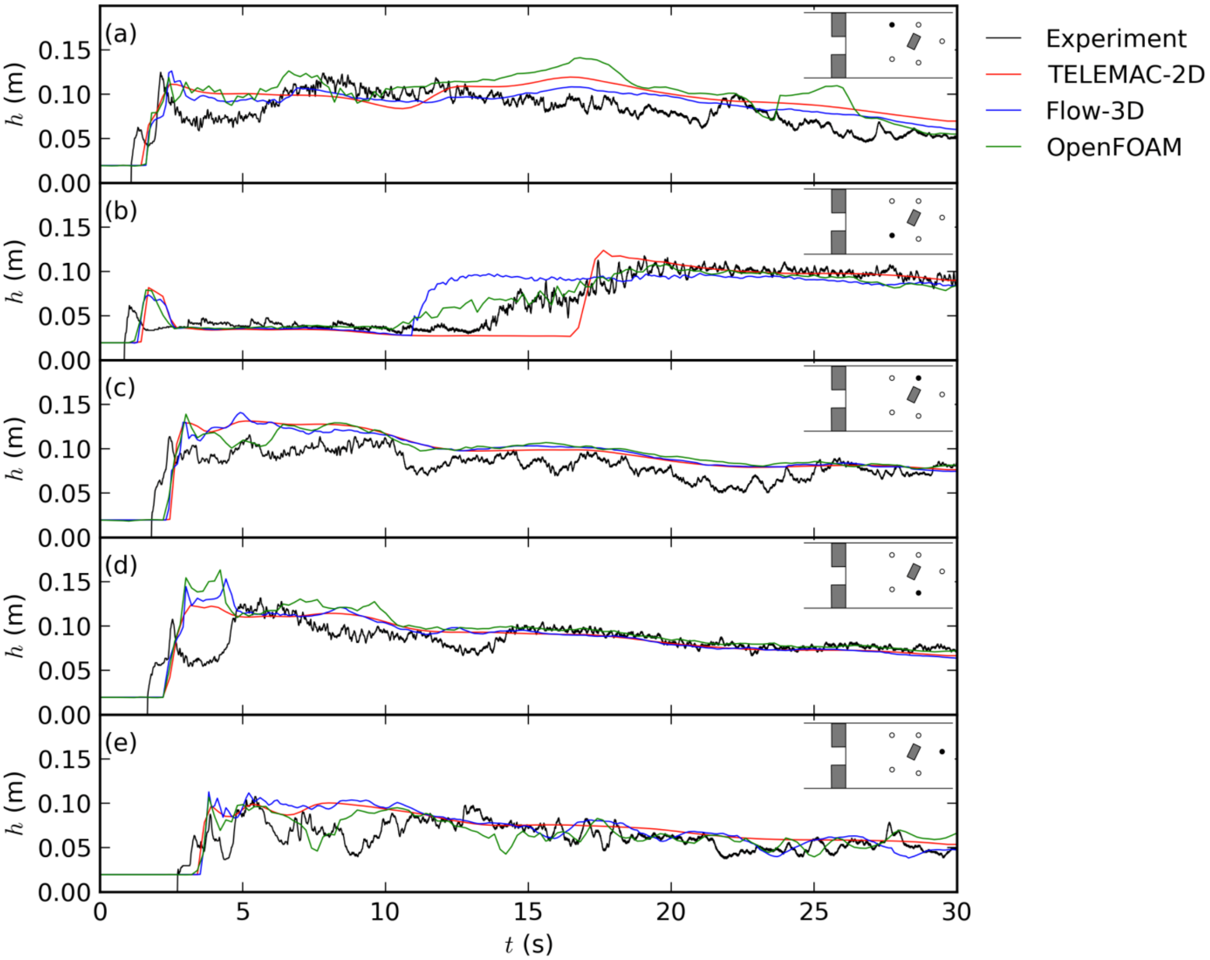

Figure 5. Comparison between experimental and numerical results: water level time series at (a) gauge G1, (b) gauge G2, (c) gauge G3, (d) gauge G4, and (e) gauge G5.

The vortex shedding in the obstacle's wake produces an oscillating water level at gauge G5, which is best captured by OpenFOAM (Figure 5 and Table 2). However, OpenFOAM is not consistently better than FLOW-3D or TELEMAC-2D at all gauges. In fact, it produced the highest RMS error at gauges G1 and G4 (Table 2). The modelled water levels are generally within 2 cm of the measured values and, on average, the models perform equally well. This result confirms that all three models can accurately simulate dam-break waves in the presence of an obstacle.

Table 2. Error Analysis: *Dam-break flow against an isolated obstacle*.

Gauge	**RMS Error (cm)**		
	TELEMAC-2D	**FLOW-3D**	**OpenFOAM**
1	2.1	1.7	2.6
2	1.8	2.1	1.3
3	2.3	2.1	2.0
4	2.0	2.1	2.4
5	1.9	2.0	1.7
Average	2.0	2.0	2.0

4. SUMMARY AND CONCLUSIONS

The three models tested in this paper, TELEMAC-2D, FLOW-3D and OpenFOAM, successfully reproduced the flood waves caused by sudden dam breaks in two independent experiments where obstacles were present. Accurately reproducing the dam-break waves in the isolated obstacle experiment was especially challenging, but the results of the three models reported herein are notably good compared to previous results reported in other studies (Soares-Frazão et al. 2003; Neelz and Pender 2013). At one particular gauge where a moving hydraulic jump was observed, OpenFOAM provided one of the best results reported so far.

The 2D shallow-water approach is sufficient to accurately predict the water level data used in this study. At best, there is only a marginal improvement when using the fully 3D CFD models, and only at certain locations. Given that 3D models are considerably more computationally demanding, especially for field-scale simulations, the results of this study show no practical advantage of using 3D models to compute water levels for large-scale dam-break problems. However, the gauges in the experiments used for numerical model validation were located a large distance away from the dam relative to the initial water depth in the reservoir (greater than thirty-five times for the triangular bottom sill experiment and greater than six times for the isolated obstacle experiment), suggesting that the 3D effects of water rushing down at the dam section have diminished. In cases where dam-break flow details are wanted near the dam, the use of 3D models may still be beneficial.

Another possible dam-break flow application where 3D models may be needed is for computing the hydrodynamic force of a dam-break wave impacting an obstacle, for example, a powerhouse, building, or bridge pier. When a dam-break wave impacts an obstacle, the water level will abruptly rise against the obstacle, creating significant vertical acceleration that is ignored in the shallow-water approach. This could be an interesting topic for future research.

5. REFERENCES

Aureli, F., Dazzi, S., Maranzoni, A., Mignosa, P., and Vacondio, R. (2015). Experimental and numerical evaluation of the force due to the impact of a dam-break wave on a structure. *Advances in Water Resources, 76*, 29-42.

Biscarini, C., Francesco, S. D., and Manciola, P. (2010). CFD modelling approach for dam break flow studies. *Hydrology and Earth System Sciences, 14*(4), 705-718.

Chanson, H. (2009). Application of the method of characteristics to the dam break wave problem. *Journal of Hydraulic Research*, 47(1), 41-49.

Dressler, R. (1952). Hydraulic resistance effect upon the dambreak functions. *Journal of research of the National Bureau of Standards, 49*(3), 217-225.

Erpicum, S., Archambeau, P., Dewals, B., Ernst, J., and Pirotton, M. (2009). Dam-break flow numerical modeling considering structural impacts on buildings. *33rd IAHR Congress: Water Engineering for a Sustainable Environment,* Vancouver, British Columbia, Canada.

Fraccarollo, L., and Toro, E. F. (1995). Experimental and numerical assessment of the shallow water model for two-dimensional dam-break type problems. *Journal of Hydraulic Research, 33*(6), 843-864.

Frazão, S. S., and Zech, Y. (2002). Dam break in channels with 90 bend. *Journal of Hydraulic Engineering, 128*(11), 956-968.

Hervouet, J.-M., and Petitjean, A. (1999). Malpasset dam-break revisited with two-dimensional computations. *Journal of Hydraulic Research, 37*(6), 777-788.

Hirt, C. W., and Nichols, B. D. (1981). Volume of fluid (VOF) method for the dynamics of free boundaries. *Journal of Computational Physics, 39*(1), 201-225.

Hirt, C., and Sicilian, J. (1985). A porosity technique for the definition of obstacles in rectangular cell meshes. *4th International Conference on Numerical Ship Hydrodynamics*, Washington, DC, USA.

Hou, J., Liang, Q., Zhang, H., and Hinkelmann, R. (2014). Multislope MUSCL method applied to solve shallow water equations. *Computers & Mathematics with Applications, 68*(12), 2012-2027.

Hunt, B. (1984). Perturbation solution for dam-break floods. *Journal of Hydraulic Engineering, 110*(8), 1058-1071.

Issa, R. I. (1986). Solution of the implicitly discretised fluid flow equations by operator-splitting. *Journal of Computational Physics, 62*(1), 40-65.

Kim, H.-J., Lee, J. W., and Cho, Y.-S. (2010). Numerical simulation of shallow-water flow using a modified Cartesian cut-cell approach. *Journal of engineering mechanics*, 136(3), 399-404.

LaRocque, L. A., Imran, J., and Chaudhry, M. H. (2013). 3D numerical simulation of partial breach dam-break flow using the LES and k–ϵ turbulence models. *Journal of Hydraulic Research, 51*(2), 145-157.

Lobovský, L., Botia-Vera, E., Castellana, F., Mas-Soler, J., and Souto-Iglesias, A. (2014). Experimental investigation of dynamic pressure loads during dam break. *Journal of Fluids and Structures, 48*, 407-434.

Miller, S., and Hanif Chaudhry, M. (1989). Dam-break flows in curved channel. *Journal of Hydraulic Engineering, 115*(11), 1465-1478.

Murillo, J., García-Navarro, P., and Burguete, J. (2009). Conservative numerical simulation of multi-component transport in two-dimensional unsteady shallow water flow. *Journal of Computational Physics*, 228(15), 5539-5573.

Neelz, S. and Pender, G. (2013). Benchmarking the latest generation of 2D hydraulic modelling packages. Report SC120002, Environment Agency, Bristol, UK.

Oertel, M., and Bung, D. B. (2012). Initial stage of two-dimensional dam-break waves: Laboratory versus VOF. *Journal of Hydraulic Research, 50*(1), 89-97.

Ozmen-Cagatay, H., and Kocaman, S. (2010). Dam-break flows during initial stage using SWE and RANS approaches. *Journal of Hydraulic Research, 48*(5), 603-611.

Ritter, A. (1892). Die fortpflanzung de wasserwellen. *Zeitschrift Verein Deutscher Ingenieure, 36*(33), 947-954.

Shigematsu, T., Liu, P. L.-F., and Oda, K. (2004). Numerical modeling of the initial stages of dam-break waves. *Journal of Hydraulic Research, 42*(2), 183-195.

Soares-Frazão, S. (2007). Experiments of dam-break wave over a triangular bottom sill. *Journal of Hydraulic Research, 45*(sup1), 19-26.

Soares-Frazão, S., Noel, B., Spinewine, B. and Zech, Y. (2003). The isolated building test case: results from the IMPACT benchmark. *3rd IMPACT project workshop*, Louvain-la-Neuve, Belgium.

Soares-Frazão, S., and Zech, Y. (2007). Experimental study of dam-break flow against an isolated obstacle. *Journal of Hydraulic Research, 45*(sup1), 27-36.

Toro, E. F. (2009). *Riemann solvers and numerical methods for fluid dynamics: a practical introduction*: Springer Science & Business Media.

Vasquez, J., and Roncal, J. (2009). Testing River2D and FLOW-3D for Sudden Dam-Break Flow Simulations. *Canadian Dam Association Annual Conference*, Whistler, BC, Canada.

Yu, C., and Duan, J. (2012). Two-dimensional depth-averaged finite volume model for unsteady turbulent flow. *Journal of Hydraulic Research, 50*(6), 599-611.