DETAILED MEASUREMENTS OF A DAM-BREAK FLOW. NUMERICAL SIMULATION WITH FLUENT® SOFTWARE

R. Aleixo[1], L. Larocque[2], J. Imran[2], M.H. Chaudhry[2], Y. Zech[1] & S. Soares-Frazão[1,3]

[1]IMMC - Institute of Mechanics Materials and Civil Engineering , Université Catholique de Louvain, Belgium
Place du Levant, 1 B-1348 Louvain-la-Neuve

[2]Dept. of Civil and Environmental Engineering, University of South Carolina, USA
300 Main St. Columbia, SC 29205

[3]Fonds de la Recherche Scientifique-FNRS Belgium B-1000 Bruxelles

E-mail: rui.aleixo@student.uclouvain.be

Abstract

Assessing the risk caused by the breach of a future or existing dam is a major challenge in fluvial hydraulics. The flow following a dam break is an example of a severe transient flow where both water surface and velocity change very rapidly, both in space and time. It is possible to study this flow experimentally but the limitations of the existing experimental techniques and the associated costs do not allow large scale studies. Usually the dam-break is modelled by shallow-water models that have several limitations, namely the closure equation for the shear stress and also the fact that the turbulence is seldom accounted for. To overcome the limitations of the shallow-water model, models based on the solution of the Navier-Stokes equations may be used. In this paper, a simple case of dam-break flow is analyzed with the commercial software FLUENT® and the computed and experimental results are compared.

1. Introduction

The dam-break flow is an example of severe transient flow. This flow can have drastic consequences for the human activities downstream of a dam. Several failures of the recent past resulted in significant losses in human lives and property damages (Chanson, 2004), so demonstrating the importance of studies about dam break waves.

A sequence of images of a laboratorial dam-break flow is shown in Figure 1 where drastic variation of the water level both in time and space is visible. Being a gravity-driven flow the variables that control the flow are the initial water depth upstream of the gate figuring the dam, h_0, and the gravity acceleration, g.

The dam-break wave has been studied since the end of the 19th century namely by Ritter (1893), who proposed a theoretical solution for the dam-break flow over a flat smooth bottom. The water height evolution and the depth-averaged velocity can be expressed in non-dimensional form as:

$$Z = \left(\frac{1}{3} \left(2 - \frac{X}{T} \right) \right)^2 \quad (1)$$

$$U = \frac{2}{3} \left(1 + \frac{X}{T} \right) \quad (2)$$

where $X = x/h_0$, $Z = z/h_0$, $T = (h_0/g)^{0.5}$.

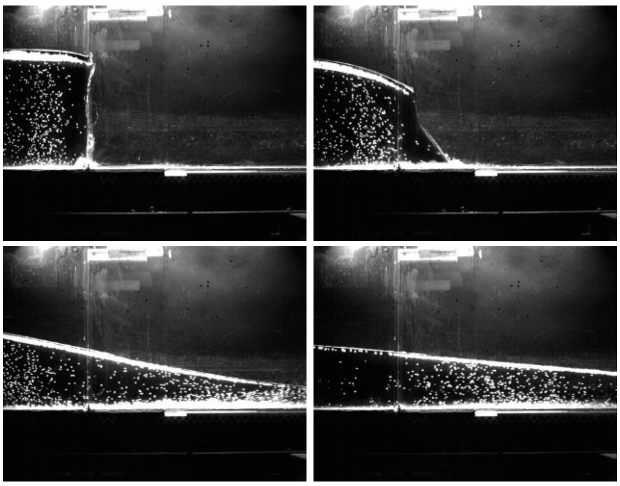


Figure 1: Different stages of the dam-break flow: t = 0 s, b) t = 0.10 s, c) t = 0.5 s d) t = 3 s.

Dressler (1952) and Whitman (1955) extended the Ritter solution to account for friction. Dressler (1952) considered that only the wave front was responsible for the friction and, by means of the boundary-layer theory he introduced a correction to the Ritter (1893) solution. Nevertheless, both the Ritter and the Dressler solutions are too simplistic to be used in real cases. The lack of theoretical expressions that can be applied to real cases motivated the development of numerical models where the flow equations are solved.

Recent numerical models allow also modelling the bed erosion by means of the Exner equation coupled with flow equations (Zech et al. 2009).
Most of the models are usually shallow water models, based on the St.-Venant equations; their applicability has been tested with relative success in many cases. But the fact that these models are often based on the shallow-water assumptions makes these models quite limited, for example, in simulating the vertical distribution of velocity. On the other hand, the closure equation for the shear-stress that is often based on Chézy-like equations derived for steady flow, are of questionable application in a severe transient flow such as a dam-break. The lack of turbulence modelling is another limitation which is often present in such models. To overcome these limitations, the authors investigated the application of FLUENT®, a Reynolds-averaged Navier-Stokes (RaNS) equations based model, to simulate the dam-break flow. FLUENT® is a commercial code computational Fluid Dynamics code, widely used in industry that can overcome the limitations of the shallow-water based models. Although FLUENT® has been widely used for industrial applications, it is necessary to validate this numerical model in the dam-break case. The numerical results for the dam-break flow are validated by using experimental results measured in the laboratory using modern measurement techniques such as Particle Tracking Velocimetry (Tropea et al. 2009).

2 Numerical Model

Dam-break flows are extremely fast and it is very difficult to get data during an experiment by using traditional methods. However, the particle tracking method allows for precise measurement due to fast acquisition camera used. A two-dimensional mesh was generated using grid-generating software, GAMBIT (Gambit, 2006), for the dam-break laboratory experimental set-up and defined to have two separate areas: water and air. The scheme shown in the Figure 2 indicates the initial conditions in the flume, before the gate lift, simulating the dam-break.

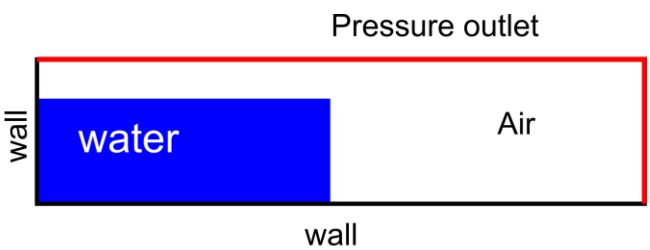


Figure 2: Representation of the computational domain.

The mesh was indicated as a Volume of Fluid (VOF) Model, which is a surface tracking technique where the interface between the two immiscible fluids, water and air, are the place of interest (Fluent, 2006). For the Pressure-Velocity Coupling, PISO was utilized for transient flow simulations (Fluent, 2006). Regarding the turbulence, the $k-\varepsilon$ model was used. Launder, et al. (1974) developed the Standard k - ε model to improve the mixing length model, this model has been shown to be useful for free-shear layer flows with relatively small pressure gradients (Launder & Sharma, 1974).
On the other hand, the Volume of Fluid (VOF) formulation relies on the fact that two or more fluids remain separate. The VOF model is a surface-tracking technique applied to a fixed Eulerian mesh. It is designed for two or more immiscible fluids where the position of the interface between the fluids is of interest (Fluent, 2006). This method was proposed by Hirt and Nichols (1981). In the VOF model, a single set of momentum equations is shared by the fluids and the volume fraction of each of the fluids in each computational cell is tracked throughout the domain. Applications of the VOF model include stratified flows, free-surface flows, filling, sloshing the motion of large bubbles in a liquid, the prediction of jet break-up and the steady transient tracking of any liquid-gas interface (Hirt and Nichols, 1981) and also the motion of liquid after a dam-break. For example, Mohapatra et al. (1999) used a variation of the VOF method to study the dam-break flow in a vertical plane. Ling et al. (2001) used the VOF method to determine the curved free surface in open channel flows. The VOF method was also used by Chen et al. (2002) and by Anderson et al. (2010) for the study of different spillways. A dam-break study of the initial stages of the dam-break flow was proposed by Oertel and Bung (2012).

2.1 The Dam-break Channel and Mesh

The simple problem geometry allows for efficient discretisation of the domain using quadrilateral cells. In Abdolmaleki, et al. (2004), an investigation of a VOF dam-break grid cell size was compared. They examined different grid sizes to analyze the sensitivity and accuracy of the results. The results showed that refining an uniform grid increased the computational time and degraded the results. Therefore, with a higher number of cells, the computational time increased without an improvement in the results.
For this simulation, the grid cell size is 0.001 m. The channel is 6 m long, 0.25 m wide and 0.5m high. The boundary conditions were defined to match the experimental inputs. The sides, back (upstream), sides surrounding the outlet, and bottom were specified as rigid impermeable walls. The top of the flume and downstream outlet are defined as pressure outlets. Pressure outlet boundary conditions require the specification of static pressure at the outlet (Fluent, 2006).
When an unsteady VOF calculation is performed in FLUENT® (Fluent, 1997), a time step different from the

one used for the rest of the transport equations is defined for the volume fraction calculation. The time step is refined based on the input for the maximum Courant number, *Cr* allowed near the free-surface. For this simulation the Courant number used was $C_r = 0.25$. A proper time step is a function of the grid size and initial water depth. The velocity and time step have to comply with the CFL condition:

$$C_r = \frac{\Delta t}{\Delta x_{cell}} V \leq 1 \tag{3}$$

Where Δt is the time step, Δx is the grid size and V is the maximum velocity of the flow. With an upstream head, $h_0 = 0.325$ m, and the grid size, $\Delta x = 0.004$, the maximum velocity of the dam-break flow was estimated by $V = (2gh_0)^{0.5} = 2.52$ m/s leading to a time step of 0.0016 s.

3 Experimental Set-up and Measurement Techniques

In order to validate the above numerical model, dam break experiments were performed in the dam break channel of the Hydraulics Laboratory of Université catholique de Louvain. The channel has a rectangular section of 0.25 m with by 0.5 m height and its total length is 6 m. The side walls are made of glass to allow visual observation of flow. The reservoir bottom is made of polished impervious wood and the test section bottom is made of glass. In Figure 3 a photograph and a scheme of the channel are presented.

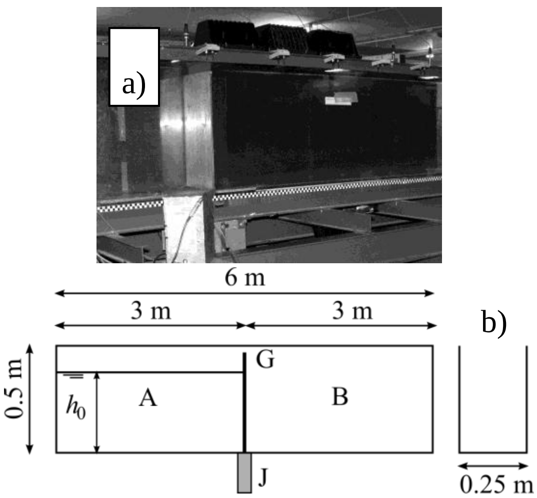


Figure 3: a) The dam-break channel in the lab. b) Schematics of the channel. J: pneumatic jack. A: reservoir; B test section, G: gate.

A gate placed at the middle of the channel divides it into two equal reaches of 3 m each, the upstream one being the reservoir and the downstream one the test section, as indicated in Figure 3. The gate G is pulled downward by a pneumatic jack J which allows a removal time of about 120 ms, which complies with the criterion put forward by Lauber and Hager (1998) for the considered initial water height, $h_0 = 0.325$ m:

$$T_r = 0.12 \text{ s} < \sqrt{2\frac{h_0}{g}} \approx 0.25\text{s} \tag{4}$$

The intrinsic features of the dam-break flow limit the set of measurement techniques to be used (Aleixo et al. 2010). Imaging methods are therefore preferable therefore preferable allowing both high temporal and spatial resolutions. To extract the velocity information from flow images, two main methodologies are available: one is the Particle Image Velocimetry (PIV) (Raffel et al. 2007) and the other is the Particle Tracking Velocimetry (PTV) (Capart et al. 2002, Tropea et al. 2009). In this paper the velocity is obtained by means of PTV. The PTV algorithm used is based on the Voronoï tessellation of space, using the detected particles' positions as the centre of a Voronoï polygon, as proposed by Capart et al. (2002), who develop it for granular flow. For application to dam-break flow see Aleixo et al. (2011). Images for PTV were acquired by means of a DALSA camera with 1 Mpix resolution and 100 Hz acquisition frequency.

The particles used in the experiments are made of high-density polystyrene (PEHD) with a density of about 0.97. They are coated with a white film in order to improve light reflection. The flow is lighted by means of three 1000 W light spots. In Figure 4 a raw image of the flow taken at $T = 1$ and the corresponding velocity field are shown.

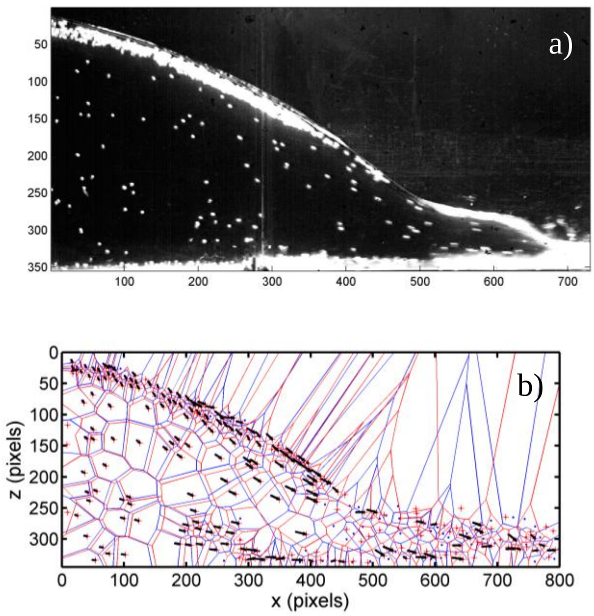


Figure 4: a) Raw image of the flow at T=1. b) Voronoï Tesselation and velocity field for T =1.

4. Numerical Model Validation

The validation of the numerical model was made by means of the experimental results obtained for the water level and velocity profiles. Two non-dimensional time were chosen $T = 1$ ($t = 0.182$ s) and $T = 5$ ($t = 0.91$ s), corresponding to the near-field and far-field behaviour respectively.

4.1 The Volume of Fluid (VOF) Method

The use of the VOF method allows tracking the free surface. In Figure 5a and 5b the water body is identified and plotted against the water surface profile obtained from the images of the flow. It can be seen that for $T = 1$ the computed water wave is less developed than the observed one, while for $T = 5$ the agreement is fairy good.

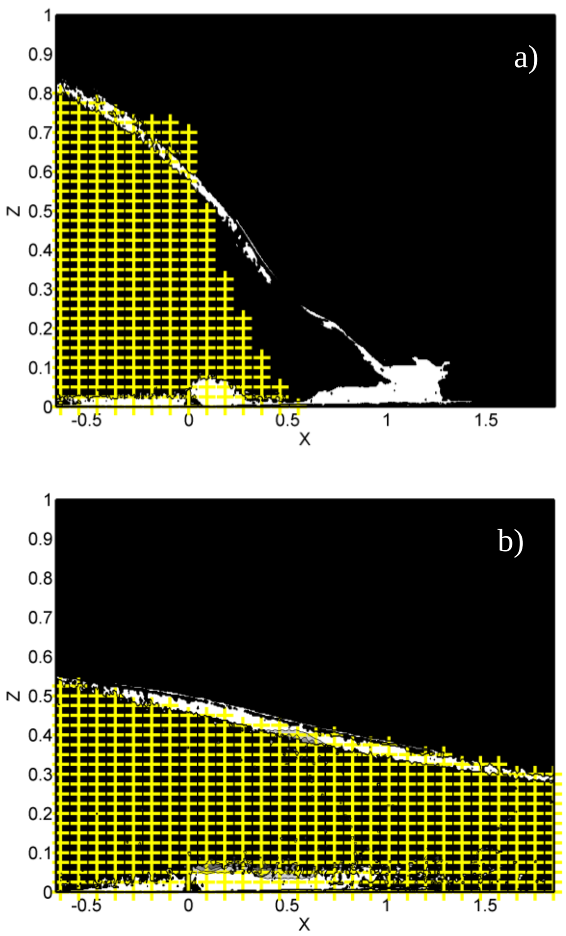


Figure 5 Comparison of water wave evolution against numerical results (+). a) $T = 1$; b) $T = 5$.

4.2 Velocity profiles

In Figure 6 the computed velocity field of the considered dam-break flow is shown for $T = 1$ and $T = 5$.

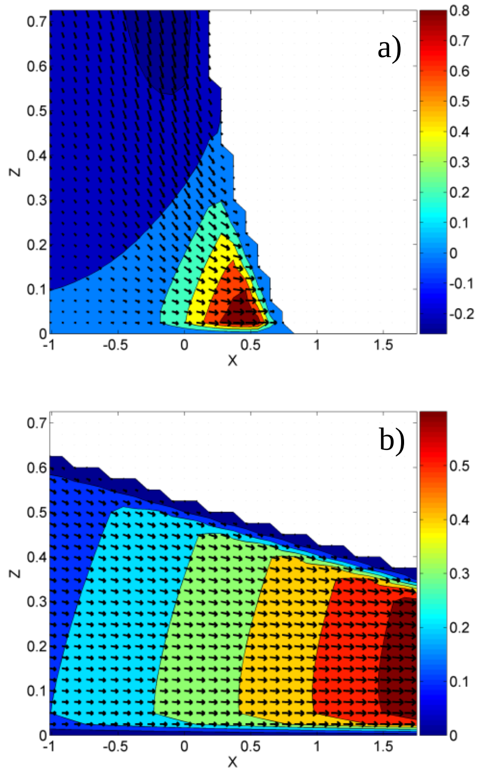
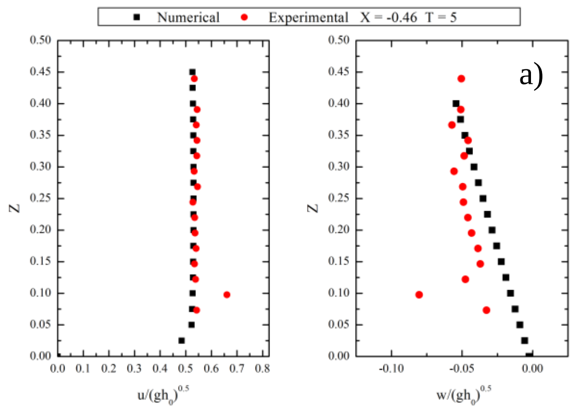


Figure 6: Velocity field for the dam-break flow calculated by FLUENT® for a) $T = 1$ and b) $T = 5$. The color maps show the non-dimensional modulus of the velocity: $|\mathbf{v}|/c_0 = ((u^2+w^2)^{0.5})/c_0$

The velocity profiles obtained with FLUENT® were compared with the ones measured with the Voronoi PTV technique. For $T = 5$ the results are shown in Figure 7 for 4 different sections, $X = -0.46$, -0.09, 0.27 and 0.67.



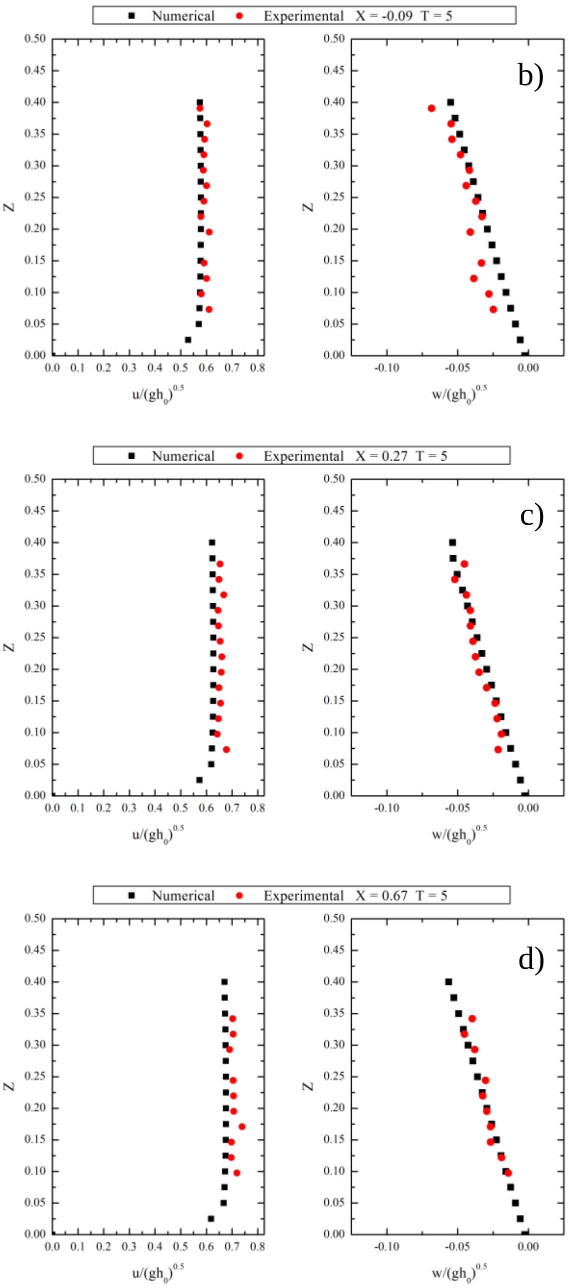


Figure 7: Numerical and experimental velocity profiles measured at $T = 5$ at different sections, a) X = – 0.46, b) $X = -0.09$, c) $X = 0.27$ and d) $X = 0.67$

As it can be seen the numerical results are in fairy good agreement with the experimental ones except for the vertical velocity component, *w*, in section $X = -0.46$. The difference between experimental and numerical results is evaluated according to the expression:

$$d = \frac{\left| u_{\text{exp}} - u_{\text{num}} \right|}{u_{\text{exp}}} \times 100 \quad (\%) \qquad (5)$$

The deviation profile between computed and observed velocity along the *Z*-coordinate is plotted in Figure 8. It can be seen that the differences for the horizontal component are usually less than 10 % whereas the differences for the vertical component are quite high, which was expected as the value of this vertical component is rather small.

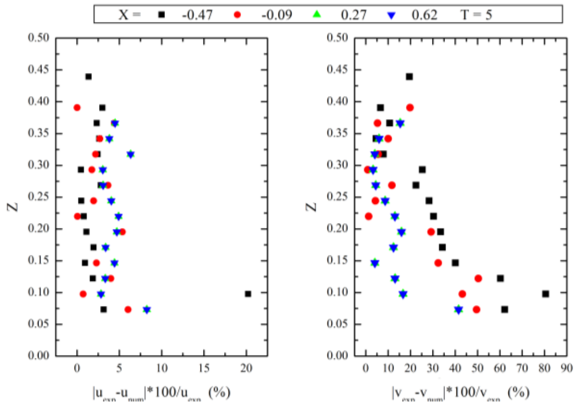


Figure 8: Differences between experimental and numerical results and their evolution along *Z*.

5 Conclusions

The commercial software for Fluid Dynamics, FLUENT® was used to compute the dam-break flow over a smooth bed in order to obtain the flow height evolution and the velocity field. The obtained results were compared with the experimental results obtained for a dam-break flow in similar conditions and measured with imaging techniques. Water height evolution compares better for the later times than for the first instants. A reason for that might be the way the gate mechanism works which is not taken into account by FLUENT®. The obtained velocity profiles compare well with the experimental values measured especially for the horizontal component, where differences of less than 10 % are registered. For the vertical component, that value is, in proportion, significantly higher.

The validation of the FLUENT® code allows one to investigate in an easier way the features of the dam-break flow into more complex cases.

Acknowledgments

The first author thanks the Foundation for Science and Technology, from Portugal for the PhD scholarship SFRH/BD/36023/2007. Funding from the US National Science Foundation PIRE Program (Grant No. 0730246) is also acknowledged.

References

Abdolmaleki, K., Thiagarajan, K.P. & Morris-Thomas, M.T. (2004). Simulation of The Dam Break Problem and Impact Flows Using a Navier Stokes Solver. 15th Australian Fluid Mechanics Conference.

Andersson, A.G., Lundström K., Andreasson, P. & Lundström T.S. (2010). Simulation of Free Surface Flow in a Spillway with a rigid lid and Volume of Fluid Methods and Validation in a Scale Method. Proceedings of the 5th European Conference on Computational Fluid Mechanics. ECCOMAS Conference, Lisbon, Portugal.

Aleixo, R. Soares Frazão, S. Spinewine, B. & Zech, Y. (2010). Velocity profiles in dam-break flows: Water and sediment layers, In: Braunschweig, Germany, pp.533-540 (Vol. 1).

Aleixo, R., Soares-Frazão, S. & Zech, Y. (2011). Velocity-field measurements in a dam-break flow using a PTV Voronoï imaging technique. Experiments in Fluids, Vol. 50. No. 6, 1633-1649.

Capart, H., Young, D.L. & Zech Y. (2002). Voronoi imaging methods for the measurement of granular flows. Exp Fluids 32: 121-135.

Dressler, (1952) R.F. Hydraulic Resistance Function upon the Dam-Break Functions. Jl. Of Research, Ntl. Bureau of Standards, vol 49, No. 3. pp. 217-225.

Chansom, H. (2004). Environmental Hydraulics for Open Channel Flows. Butterworth-Heinemann. Oxford.

Chen, Q., Dai, G. & Liu, H., Volume of Fluid Model for Turbulence Numerical Simulation of Stepped Spillway Overflow, Journal of Hydraulic Research, 128, 683-688.

FLUENT. Fluent 6.1 user manual. Fluent, 2006.

FLUENT, Chapter 10-VOF Free Surface Model. Fluent Inc. May 10, 1997.

Gambit. Gambit user manual. Gambit, 2006.

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.

Lauber, G. & Hager, W.H. (1998) Experiments to dambreak wave: horizontal channel. Journal of Hydraulic Research, 36: 291-307.

Launder, B.E. & Sharma, B.I. (1974). Application of the energy dissipation model of turbulence to the calculation of flow near a spinning disc. Letters in Heat and Mass Transfer, pages 1: 131-138.

Lin, L., Yongcan, C. & Yuliang, L. (2001) Volume of Fluid (VOF) Method for Curved Free Surface Water Flow in Shallow Open Channel, Proceedings of the XXIX IAHR Conference, Beijing, China.

Mohapatra, P.K., Eswaran, V. & Murty Bhallamudi S. (2001) Two-dimensional analysis of dam-break flow in vertical plane. Journal of Hydraulic Engineering vol. 125(2), pp. 183-192.

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

Raffel, M., Willert, C., Wereley, S. & Kompenhans, J. (2007) Particle Image Velocimetry – A Practical Guide, 2nd Edition, Springer-Verlag, Berlin.

Ritter, A. (1892) Die Fortpflanzung der Wasserwellen. Zeitschrift Verein Deutscher Ingenieure, Vol. 36, No. 2, 33, 13 Aug., pp. 947-954.

Whitman, G.B. (1955) The Effects of Hydraulic Resistance in the Dam-Break Problem. Proc. Roy. Soc. Of London. Ser. A, vol 227, pp. 399-407.

Tropea, C., Yarin, A, & Foss, J. (2007). Handbook of Experimental Fluid Mechanics. Springer. Berlin Hilderberg.

Zech Y., Soares-Frazão, S., Spinewine, B., Savary, C., & Goutière, L. (2009). Inertia effects in bed-load transport models. Canadian Journal of Civil Engineering 36:1587-1597.