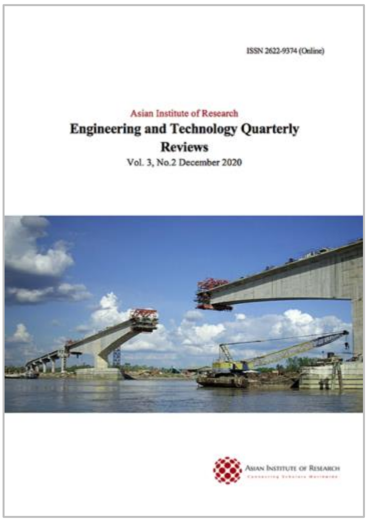


Engineering and Technology Quarterly Reviews

Syamsuri. (2020), OpenFOAM Based Approach for the Prediction of the Dam Break with an Obstacle. In: *Engineering and Technology Quarterly Reviews*, Vol.3, No.2, 91-97.

ISSN 2622-9374

The online version of this article can be found at:
https://www.asianinstituteofresearch.org/

Published by:
The Asian Institute of Research

The *Engineering and Technology Quarterly Reviews* is an Open Access publication. It may be read, copied, and distributed free of charge according to the conditions of the Creative Commons Attribution 4.0 International license.

The Asian Institute of Research *Engineering and Technology Quarterly Reviews* is a peer-reviewed International Journal. The journal covers scholarly articles in the fields of Engineering and Technology, including (but not limited to) Civil Engineering, Informatics Engineering, Environmental Engineering, Mechanical Engineering, Industrial Engineering, Marine Engineering, Electrical Engineering, Architectural Engineering, Geological Engineering, Mining Engineering, Bioelectronics, Robotics and Automation, Software Engineering, and Technology. As the journal is Open Access, it ensures high visibility and the increase of citations for all research articles published. The *Engineering and Technology Quarterly Reviews* aims to facilitate scholarly work on recent theoretical and practical aspects of Education.

Asian Institute of Research
Connecting Scholars Worldwide

The Asian Institute of Research
Engineering and Technology Quarterly Reviews
Vol.3, No.2, 2020: 91-97
ISSN 2622-9374

OpenFOAM Based Approach for the Prediction of the Dam Break with an Obstacle

Syamsuri[1]

[1] Department Mechanical Engineering, Institut Teknologi Adhi Tama Surabaya, Indonesia.
Email: syamsuri@itats.ac.id

Abstract
The phenomenon of the flow impact on a vertical wall resulting from a dam problem is simulated by using OpenFOAM. In this simulation, a dam break was also simulated with the addition of obstacles with various dimensions. The aim of this study is to assess the accuracy of the solver for problems in the impact wave category from the experimental results of previous researchers and other numerical solution techniques compared with the results of this solver. Different aspects of flow such as free surface elevation before and after the initial impact have been observed in depth. The method used in this research is numerical computation simulation with the OpenFOAM approach which has the advantage of being more accurate and fast simulation time. The variations in the dimensions of the obstacle in this study were b / h = 0.25, b / h = 0.5 and b / h = 1.0. From the simulation data, it is found that the numerical approach has been validated through quantitative comparisons with experimental measurements. The computational positions of the leading edge of the collapsed water column match the experimental data. The difference between the experiment and this numerical solution is below 2%.

Keywords: Dam Break, An Obstacle, Surface Profile, OpenFOAM

1. Introduction

Dam is a structural building that functions to accommodate and store a number of fluids (water), as well as channeling the needs for drinking water, electricity generation, and irrigation. Fluid movements (water) on dams such as heaving and rolling which trigger high pressure causes structural damage or what we know as dam break. This break damage causes considerable disasters such as floods or other natural damage. Numerical modeling studies so interesting to learn to understand the dynamics of fluid movement in this case.

Several studies with experimental methods on dam break have been carried out by previous researchers, among others: Verma *et al.* (2017) has conducted research on the experimental study of the soil dam breakthrough path using the fuse plug model. The different input parameters that help in understanding the phenomenon are the temporal variation of the initiation of the breakthrough, the breakthrough width, the breakthrough depth, the discharge intensity and the peak time. This paper provides the results of laboratory research conducted using wood fuse plugs and five different types of soil.

The behavior of the breakthrough depends on the dimensions of the fuse plug, type of fill material, reservoir capacity and inflow. Researchers on dam-break flow routing in confluent channels have been conducted by Chen *et al.* (2019). Experiments were carried out on smooth, transparent, rectangular prismatic channels to study the flow of dam break under four different confluent angles. Based on the variation in water level and flow rate, as the confuent angle increases, the effect of retardation and reduction in flooding has increased. Specifically, the arrival time of the floods was delayed by about 0.91% to 21.18%, and the peak flood discharge was reduced by about 9.05% to 58.36% flow. Kocaman *et al.* (2020) conducted a study on experimental and numerical analysis of a dam-break flow through different contraction geometries of the channel. Laboratory testing was performed on a smooth rectangular duct with a horizontal dry bed for three different lateral contraction geometries. The free surface profile and time variation of the water level in the selected area, obtained directly from three synchronized CCD video cameras, record via a virtual wave probe. The experimental results were compared with numerical solutions of volume of fluid based on shallow water equations and reynolds-averaged navier-stokes (RANS) with the k-ε turbulence model. A good agreement is obtained between the computed results and the measurement results. The new experimental data presented can be used to validate numerical models for simulating dam flows over irregular topography.

Numerical modeling is an option because it is effective, safe and cost-effective. Various methods for the study of numerical modeling of fluid motion dam break has been applied. By using a method LS/IB, Yu *et al.* (2017) able to show similar results between the experimental and numerical modeling of fluid motion at any given time. On a different method, Issakhov *et al.* (2019) were able to show the movement of fluid turbulence at any particular time and a different angle obstacle by using VOF method. Other than that, Yu *et al.* (2019) using the CLSVOF / IB method to simulate the movements fluid that are affected by Reynold's number. With increasing numbers of Reynolds, fluid movements tend to be unstable or turbulent. Zhao *et al.* (2017) have conducted research on numerical simulations of dam breaker floods with MPM. Dam-break flows with different initial aspect ratios are simulated in the material point method and shallow water equation models which are extensively verified. So to test the accuracy and stability of the point method material, a dam-break flow simulation is performed and the results are in accordance with the validation of other numerical methods and experimental data. The material point method shows its potential to tackle the hydrodynamics like this case.

OpenFOAM software is a type of software development from CFD. This software is widely used in industry because it is open source and can be modified. For modeling, this software is able to simulate the movement of two-phase fluids with a specific turbulence model by Higuera *et al.* (2013). So that in this study the authors used software OpenFOAM 2.2.2. to simulate the dam break with variation ratio of obstacle, because this topic is still not widely discussed.

2. Method

In this study using the application of breaking of dam. The geometry is shown in Figure 1 where the boundaries are the walls with the top open. Pressure on the surface is set at 1 ATM. The fluid used in this simulation is water with kinematic viscosity of $1.0 \times 10-6\ m2.\ s-1$ and a density of $1.0 \times 103\ kg.\ m-3$. The fluid was dropped naturally, so that the fluid moves due to the acceleration of gravity at $9.81\ m.\ s-2$ which leads down. At the bottom of the middle side as shown in Figure 1 there is an obstacle, which in this simulation will be varied with the ratio of width and height of the obstacle. The width of the obstacle in the set remains at 0.024 m while the height of the obstacle is varied by b/h ratio of 0.25, 0.5, and 1. The movement of this water flow will later be monitored at certain times including 17 s where this condition represents the time the fluid starts to flow, 47 s represents the state of the fluid when it hits the obstacle, 58 s represents the state of the fluid some time after hitting the obstacle, and 152 s when the fluid has passed through the obstacle and hit the back wall. From the four monitoring times, flow characteristics will be analyzed at each height variation from the obstacle. In this simulation using the free surface modeling method or commonly known as the Volume of Fluid Method (VOF). Where this method in this simulation is used to calculate the motion of flux, which must be settled separately.

Mathematic Model of Flow

Laminar flow is a fluid that has a certain viscosity and it is an incompressible flow can be described by the Navier-Stokes equations in an Eulerian reference frame [Ferziger & Peric, 2002],

$$\rho\left(\frac{\partial u_i}{\partial t}+u_j.\frac{\partial u_i}{\partial x_j}\right)=\rho F_i+\frac{\partial \sigma_{ij}}{\partial x_j} \tag{1}$$

$$\frac{\partial u_i}{\partial x_j}=0 \tag{2}$$

where,
u_i is a velocity component
ρ is a density
F_i is a gravity component
$\partial\sigma_{ij}$ is a stress tensor

While the equation for stress tensor can be seen in the following equation:

$$\sigma_{ij}=-P\delta_{ij}+\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)$$

(3)

Where,
μ is a dynamics viscosity
p is a pressure
δ_{ij} is a cronecker delta

The method is based on the use of a fractional function C. Derivative of the fractional function must be equal to zero [10]:

$$\frac{\partial C}{\partial t}=V.\nabla C+\nabla.[C(1-C)U_r]=0 \tag{4}$$

where:
C = Fractional Function (C = 0 If the cell is empty, C = 1 If the cell is full)
V = fluid velocity
U_r= an artificial force that “compresses” the region under consideration

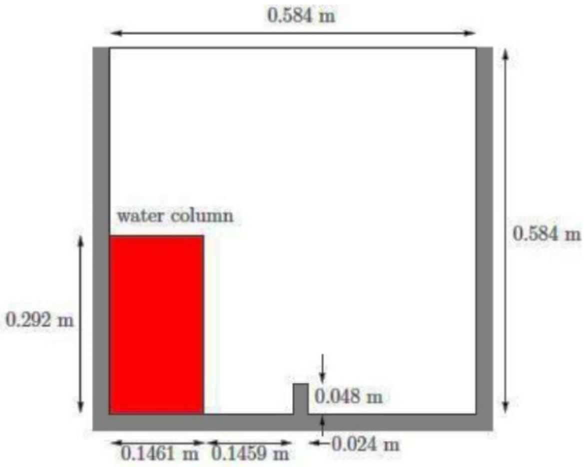


Figure 1: Dam break geometry

3. Results & Discussion

3.1 Validation for Surface Profile

Comparison of the surface profile of the time evolution of dam break between numerical and experimental studies by Kaceniauskas, 2005 when t = 0.275 sec is shown in Figure 2. This figure shows that the numerical results are in a good agreement with the experimental study results. In general, the results show that the mean error between numerical and experimental data is less than 2 %.

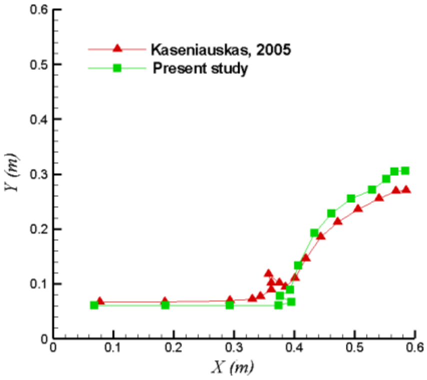
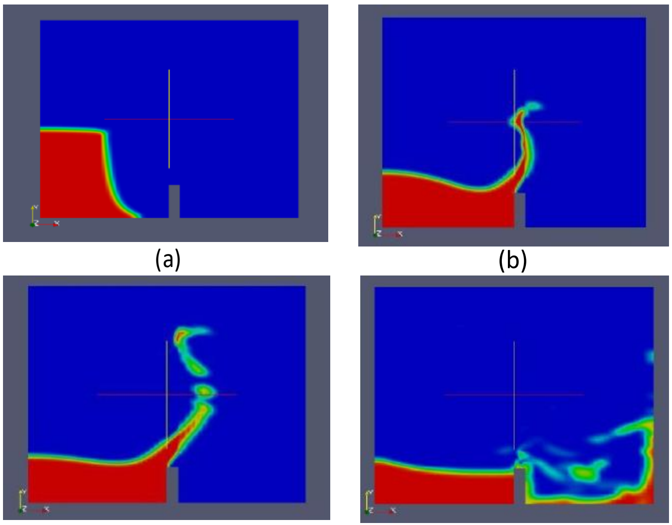


Figure 2: Surface profile of dam break with t = 0.275 sec

3.2 The phenomenon of flow characteristics of obstacle with ratio b / h = 0:25 at 17 s, 47 s, 58 s, and 152 s

The characteristic of the dam break flow phenomenon which is affected by time changes is shown in Figure 3. When the floodgate is opened, the water will flow. This happens when t = 17 s shows the flow of water moves to fill the empty space on the left bottom wall of the obstacle (figure 3a) caused by the existence of gravity. Then when t = 47 s shows the flow of water completely hits the barrier wall and moves in the vertical direction with the formation of a curve on the surface (figure 3b). The curvature on the surface occurs because water experiences an inertia force [Zhainakov & Kurbanaliev, 2013]. Furthermore, when t = 58 s shows the formation of a bubble at the end of the flow (figure 3c). When the gravitational force is greater than the inertia force, the vertical flow moves downward and hits the bottom right bottom wall of the obstacle ([Biscarini *et al.*(2010)] , [Oertel & Bung, 2012]). Furthermore, the flow will move upwards and there will be bubbles around the wall (figure 3d). This bubble formation caused by interactions between water and air ([Hansch *et al.* (2014)], [Ryu *et al.*(2007)]).



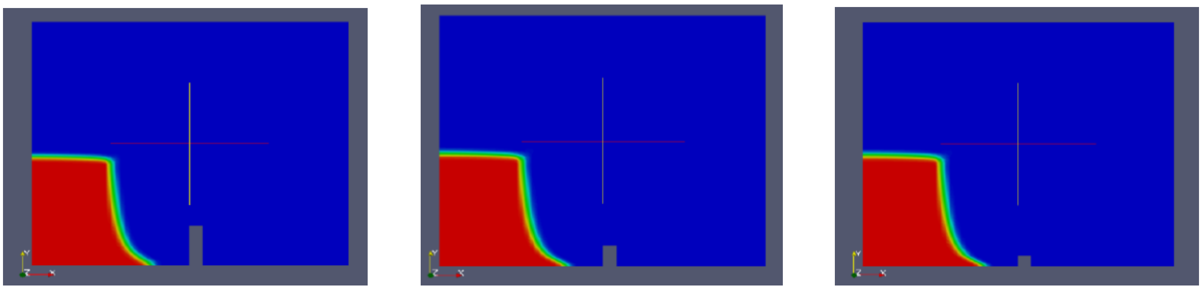
(a) (b)

(c) (d)

Figure 3: The phenomenon of flow characteristics of obstacle with ratio b/h = 0.25 at
(a) 17 s, (b) 47 s, (c) 58 s, and (d) 152 s

3.3 The flow characteristic phenomenon at 17 s with ratio of obstacle b/h = 0.25, 0.5, dan 1

The effect of the obstacle ratio b / h on dam break flow at 17 s is shown in Figure 4. This phenomenon shows that when the water gate is opened, the flow of dam break is almost the same, namely the flow of water moves to fill the empty space on the bottom left wall of the obstacle. That is because the dam break flow has not hit the obstacle and the speed at which the fluid falls in all obstacle variations only utilizes the acceleration of gravity. So that all contours produced show the same conditions.

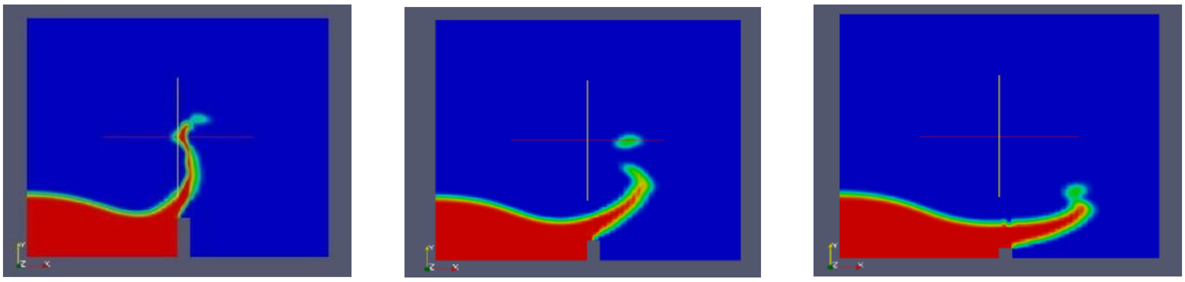


(a) (b) (c)

Figure 4: The flow characteristic phenomenon at 17 s with a ratio of obstacle
(a) b/h = 0.25, (b) b/h = 0.5, and (c) b/h = 1

3.4 The flow characteristic phenomenon at 47 s with ratio of obstacle b/h = 0.25, 0.5, dan 1

Figure 5 shows the effect of the ratio b / h to the dam break flow at 47 s. This phenomenon shows that when the obstacle b / h ratio is getting bigger it is seen that the end of the flow tends towards horizontal and when the obstacle ratio b / h gets smaller then it appears that the end of the flow tends towards the vertical. This happens because the smaller the ratio b/h obstacle produces collision pressure that occurs in the flow dam break is greater, conversely the greater the ratio b / h obstacle produces greater gravity the flow of dam break ([Issakhov & Imanberdiyeva (2019)] , [Issakhov *et al.*(2018)]).

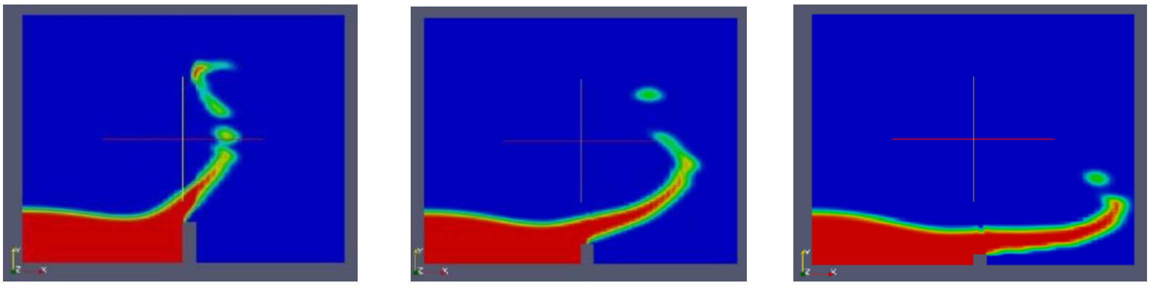


(a) (b) (c)

Figure 5: The flow characteristic phenomenon at 47 s with a ratio of obstacle
(a) b/h = 0.25, (b) b/h = 0.5, and (c) b/h = 1

3.5 The flow characteristic phenomenon at 58 s with ratio of obstacle b/h = 0.25, 0.5, dan 1

The effect of the obstacle b / h ratio on dam break flow at 58 s is shown in Figure 6. From this phenomenon when the obstacle b / h ratio is getting bigger it is seen that the end of the flow has tended towards horizontal and when the obstacle b / h ratio is getting smaller then it appears that the end of the flow tends towards the vertical ([Issakhov & Imanberdiyeva, 2019] , [ssakhov *et al.*(2018)]). The bubble at the end of the flow is caused by air trapped in the dam break flow ([Hansch *et al.* (2014)], [Ryu *et al.*(2007)]).

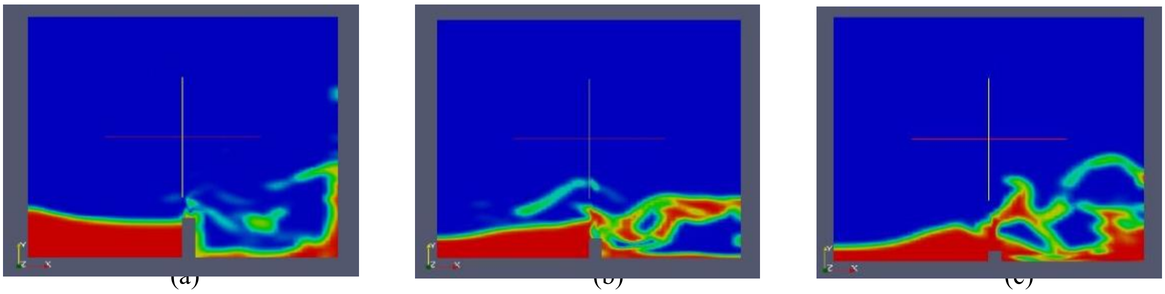


(a) (b) (c)

Figure 6: The flow characteristic phenomenon at 58 s with a ratio of obstacle
(a) b/h = 0.25, (b) b/h = 0.5, and (c) b/h = 1

3.6 The flow characteristic phenomenon at 152 s with ratio of obstacle b/h = 0.25, 0.5, dan 1

The effect of the obstacle b / h ratio to the dam break flow at 152 s is shown in Figure 7. From this phenomenon, it can be seen that the flow of dam break when passing through the obstacle b / h = 0.25, there is still a lot of water retained by the obstacle compared to other b / h ratios, this is caused by the obstacle with the ratio b / h = 0.25 having a higher height than another. So that the ability to hold water can be more than others, but the obstacle must receive a greater impact than the others. The smaller the ratio b / h obstacle causes the flow formed tends to be more turbulent [Ozmen-Cagatay & Kocaman, 2011].



(a) (b) (c)

Figure 7: The flow characteristic phenomenon at 152 s with a ratio of obstacle
(a) b/h = 0.25, (b) b/h = 0.5, and (c) b/h = 1

3.7 Comparison of characteristics of a water column

The effect of time on the height of the water movement on the variation of the ratio b / h is shown in Figure 8. From the three graphs, when the time is longer the height of the water movement is lower. The ratio b / h = 0.25 has a higher water movement height than the other b / h ratio. This indicates that the high ratio of b / h will inhibit the flow of fluid.

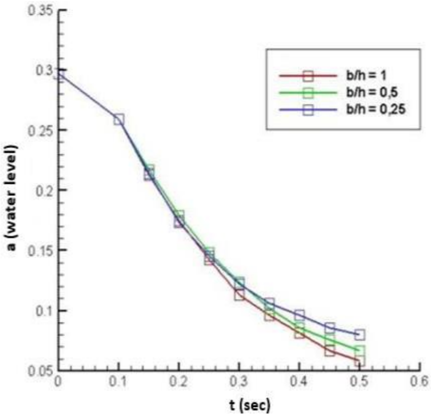


Figure 8. Effect of time on height of water movement on obstacle ratios b/h = 0.25, b/h = 0.5 and b/h = 1

5. Conclusions

Validation has been carried out with previous studies using experimental methods compared with OpenFOAM simulation results. The validation results show that, there are similarities of the surface profiles for the dam break when t = 0.275 sec. For case 3 with obstacle b / h = 1.0, when t = 17 sec the phenomenon of water flow near the lower wall has moved to the front wall but has not yet hit the obstacle, the same was the case for cases 1 & 2. At t = 47 sec the flow has hit the obstacle and the water follows the contour of the obstacle then the water moves vertically. Meanwhile, at t = 58 sec, it was seen that bubbles formed and water began to separation from the main stream. However, when t = 152 sec, it was seen that the fluid flow passing through the obstacle had hit the front wall, and the bubble was also getting bigger. The fluid flow formed at this time tends to be more turbulent.

Acknowledgments

We would like to thank to ITATS for funding this research. Special thank’s to Prof. Ming Jyh CHERN and Assistant Prof. Nima Vaziri for very useful discussions.

References

Biscarini, C., Di Francesco, S. & Manciola, P. 2010. CFD modeling approach for dam break flow studies,” *Hydrology and Earth System Sciences*, 14, pp.705-718.

Chen, S., Li, Y., Tian, Z. & Fan, Q. 2019. On dam-break flow routing in confluent channels. *Int. J. Environ. Res. Public Health*, 16, 4384, pp.1-23.

Ferziger, J.H. & Peric, M. 2002. *Computational methods for fluid dynamics*, Springer 3rd Ed., Springer-Verlag Berlin Heidelberg.

Hänsch, S., Lucas, D., Höhne, T & Krepper, E. 2014. Application of a new concept for multi-scale interfacial structures to the dam-break case with an obstacle. *Nuclear Engineering and Design*, 279, pp.171- 181.

Higuera, P. Lara, J.L. & Losada, I.J. 2013. Realistic wave generation and active wave absorption for Navier–Stokes models: Application to OpenFOAM. *Coastal Engineering, 71*, pp. 102-118.

Issakhov, A. & Imanberdiyeva, M. 2019. Numerical simulation of the movement of water surface of dam break flow by VOF methods for various obstacles. *International Journal of Heat and Mass*, Vol.136, pp. 1030-1051.

Issakhov, A., Zhandaulet, Y. & Nogaeva, A. 2018. Numerical simulation of dam break flow for various forms of the obstacle by VOF method. *International Journal of Multiphase Flow*, 109, pp. 191-206.

Kocaman, S., Güzel, H., Evangelista, S., Ozmen-Cagatay, H. & Viccione, G. 2020. Experimental and numerical analysis of a dam-break flow through different contraction geometries of the Channel. *Water*, 12, 1124, pp.1-22.

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

Ozmen-Cagatay, H. & Kocaman, S. 2011. Dam-break flow in the presence of obstacle: experiment and CFD simulation. *Engineering Applications of Computational Fluid Mechanics*, *5*(4), pp. 541-552.

Ryu, Y., Chang, K.A. & Mercier, R. 2007. Application of dam-break flow to green water prediction,” *Applied Ocean Research*, 29 (3), pp. 128-136.

Verma, D. K., Setia, B & Arora, V. K. (2017). Experimental study of breaching of an earthen dam using a fuse plug model. *IJE TRANSACTIONS A: Basics*, Vol. 30, No. 4, pp.479-485.

Yu, C.H. & Sheu, T.W.H. 2017. Development of a coupled level set and immersed boundary method for predicting dam break flows. *Computer Physics Communications*, Vol. 221, pp.1-18.

Yu, C.H. Wen, H.L., Gu, Z.H. & An, R. D. 2019. Numerical simulation of dam-break flow impacting a stationary obstacle by a CLSVOF/IB method. *Communications in Nonlinear Science and Numerical Simulation*, Vol.*79*, 104934, pp.1-14.

Zhainakov, A.Z. & Kurbanaliev, A.Y. 2013. Verification of the open package OpenFOAM on dam break problems. *Thermophysics and Aeromechanics*, Vol.20, pp. 451-461.

Zhao, X., Liang, D. & Martinelli, M. 2017. Numerical simulations of dam-break floods with MPM *Procedia Engineering*, 175, pp.133 – 140.