

Hydrodynamic Study of the Flows Caused by Dam Break Around a Rectangular Obstacle



Debasish Dutta, Lalit Kumar, Mohammad Saud Afzal,
and Prashant Rathore

1 Introduction

A dam is a boundary that stops or controls the movement of water or the underground stream. These dams control floods and provide water for irrigation, human use, navigation, aquaculture, etc. Since dams stores a huge volume of water. Therefore, it causes some of the most dangerous disasters. Dam break study is essential to forecast, characterize, and reduce threats due to dam failure, and therefore, it attracts the interest of hydraulic engineers and researchers. Dam failure is often caused by poor initial design, structural deficiencies, poor construction, or lack of maintenance [1]. The dam failure occurs because of various causes like landslides, earthquakes, heavy rainfall, or other triggering factors [2]. Floods from dam failure are typically much more dangerous than the hydrologic flood from heavy precipitations, and thus need early precautionary measures [3]. In the worst condition, complete dam structure is removed suddenly, and stored water is released rapidly to the downstream of the river valley, huge damage to wealth and life occurs from such a failure mode. Therefore, Dam break analysis is essential for managing floods. The estimation of submergence areas and the effect of flood waves is necessary for mitigation of hazards [4, 5]. Ritter (1892) was the first to study dam break analytically and derived a solution for instantons dam break [6]. He took horizontal and frictionless channels and infinite lengths of both channels and reservoirs. Later it was further studied by many researchers by changing the downstream slope and lengths [7]. Some researchers also analyzed dam break on the sloping bed [8–11]. To better understand the flows due to dam break in a laboratory experiment is a vital tool; only a few researchers have analyzed dam break experimentally due to the complexity in its setup. Yeh and Petroff (2004) performed an experimental study about the effect of dam break on a

D. Dutta (✉) · L. Kumar · M. Saud Afzal · P. Rathore
Indian Institute of Technology Kharagpur, Kharagpur, India
e-mail: debasish.dutta@iitkgp.ac.in

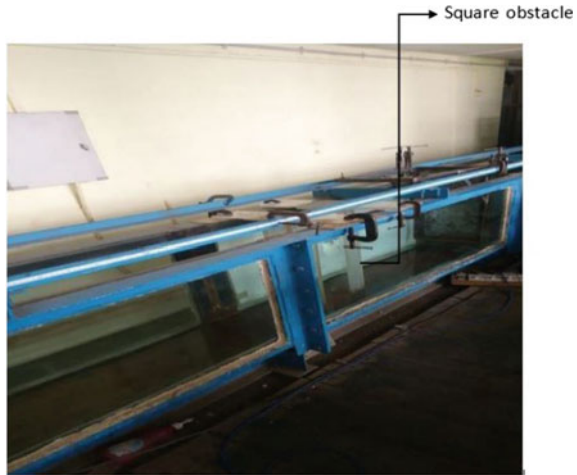
structure that has widely been used in literature and used as validation [12]. Ismail et al. (2013) obtained the velocity profile from dam break experiments and validated the results from simulation done in Ansys fluent, for measuring velocities they used Ultrasonic Doppler Velocity profilers and observe that turbulence modelling does not affect upstream velocity profile but has a significant effect at downstream of the obstacle [13]. Aureli et al. (2008) conducted an experiment, but the acquisition of data was performed through imaging techniques [14]. The test was also simulated in a 2D MUSCL-Hancock finite volume numerical model. Soares and Zech (2007) studied the dam break experimentally and obtained the changes in velocity and water level using ADV and resistive gauges [15]. Several numerical models of dam break have been developed by solving the continuity and momentum equation. Lariyah et al. (2013) analyzed dam break study for water supply projects based on the Kahang dam and later classified dam break consequences as minor, moderate, major [16]. They investigated and generated an inundation map at the downstream area and also predicted breach ow hydrograph. Feizi (2018) studied the effect of downstream obstacles (bridges and piers) caused by dam breaks on various flood patterns [17]. He used fluent 3D software for hydraulic analysis and model for free surface flow. The VOF technique was used to model the free surface. Demaio et al. (2004) also used fluent 3D software and studied the wave formed after the dam break. They modeled the wave formed at the starting of the dam break and compared the results with the actual experimental result [18]. Bai et. al. (2007) find the effect of the curvature on the dam break flows, and they found that small curvature has a minimum effect on flows due to dam break, while a greater curvature has a broad effect, causes the decrease of downstream water depth and fluctuations [19]. Yi Xiong (2011) understands dam break mechanics, peak out-flow prediction, and other essentials of dam break using the HEC-RAS model [20]. The study also predicted the submergence area and the number of villages affected due to dam failure with rehabilitation costs. The objective of the present study is to obtain the velocities magnitude upstream and downstream of the obstacle when flood waves strike the obstacle in dam break simulation in REEF3D. The simulated results are compared with experimental results for the validation of the CFD framework.

2 Experimental Set-up

The experiments have been conducted in Hydraulics and water resources engineering Laboratory, Department of Civil Engineering, Indian Institute of Technology Kharagpur. The experiments have been done in a rectangular flume of length 6 m, width 0.3 m, and height 0.6 m. The walls of the flume were made of 5 mm thick Plexiglas with metal bottom, as shown in Fig. 1.

The dam has been demonstrated by a metallic plate, hinged from top corners, and a silicone sealant is used to stop the seepage through the edges of the dam and minimize the error. The dam is located at 4 m downstream from the water's entrance and separates upstream and downstream parts of the channel. The upper

Fig. 1 Flume used for the experiment on dam break analysis



part represents the reservoir, in which water is filled up to certain different heights, and the inflow was then stopped. The water supply system comprises a constant head reservoir (dam reservoir) at an elevation of about 2 m above the ground level. Pumps are used to lift water from the underground reservoir to the dam reservoir. The constant discharge was maintained to minimize the waves formed in the storage behind the dam.

2.1 Setup of Obstacle

A square bottom obstacle has been made up of 5 mm thick Plexiglas with sides of 0.8 m, and a height of 0.6 m was placed at 0.9 m downstream of the dam. V1 and V2 are two points 8 cm upstream of the obstacle and 8 cm downstream of the obstacle where ADV is placed, and velocities at different reservoir levels are calculated. The water surface profile has been captured using a video recorder placed along the flume at different reservoir levels (i.e., $h = 20, 25, 30, 35,$ and 40). The Schematic diagram of the experimental setup with a single obstacle is shown in Fig. 2.

The Acoustic Doppler velocimeter (ADV) is an instrument used to record immediate velocity components at a point with a generally high frequency in the present study. Measurements are performed by estimating the speed of particles in a remote sampling volume dependent on the Doppler shift effect. A pulse is sent from the center transducer, and the Doppler shift presented by the reflections from particles suspended in the water is collected by the 4 (four) recipients. ADV is situated at 0.82 m and one at 1.06 m downstream of the dam (i.e., point v1 and v2). ADV tests ought to be put at least 0.05 m range from the base to take precise readings. The examining recurrence was taken to be 100 Hz.

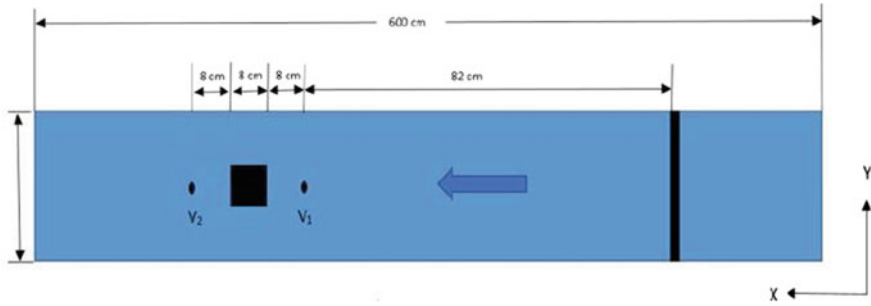


Fig. 2 Schematic diagram of the experimental setup with obstacle of top view

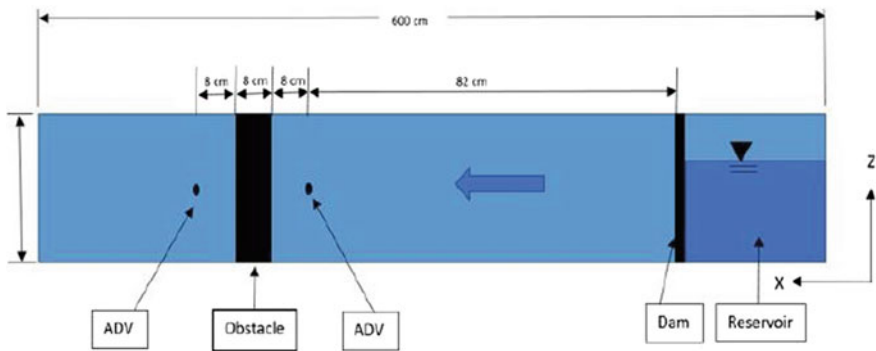


Fig. 3 Schematic diagram of the experimental setup with obstacle of side view

Figures 2 and 3 show the dam located at 4 m downstream of the inlet, water is filled up to the desired height, and inflow is stopped, waves in the reservoir are allowed to settle for some time. The channel bed is kept horizontal, then the breaking procedure was done by suddenly lifting the radial gate (t less than 98.2 s). Gate was lifted manually, and the water is allowed to pass through the flume and strike the obstacle downstream. The ADV used to measure velocity at a point is placed at point v_1 and v_2 , as shown in Fig. 2. The readings were observed by ADV and processed in a computer connected to ADV. A DSLR camera was also placed along the flume to capture the video for calculating water height, and water surface profile. Dam break Experiment was conducted for five different heights (i.e., 0.20, 0.25, 0.30, 0.35, and 0.40 m).

3 Numerical Model

The dam break analysis has drawn the interest of CFD researchers due to its (Dam) increasing size in recent years, which increases the threat to the people residing

downstream of the dam. A number of CFD techniques have been developed in the past few years. Among that, REEF3D is more user-friendly than others and also shows the flow of water in a very realistic manner [21].

3.1 Governing Equations

The fluid hydrodynamics inside the Numerical wave Tank (NWT) is solved by incompressible RANS equation along with the continuity equation as shown below.

$$\frac{\partial u_i}{\partial x_i} = 0. \quad (1)$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left[(\nu + \nu_t) \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + g_i \quad (2)$$

where u_i is the velocity of the flow, P is the pressure, water density is denoted by ρ , ν is termed as the fluid kinematic viscosity, ν_t is the eddy-viscosity, and g is the gravitational acceleration.

The dimension of the model is 10.0 m long, 0.30 m wide, and 0.60 m deep, as shown in Fig. 3. The dimension of the model's geometry is saved in a file named a control file, which can be seen in the ParaView. When the simulation is completed, the simulation results can be obtained by using ParaView software. A wake zone can be noticed at the downstream side of the obstacle.

4 Results and Discussion

During the experiment the flow changes rapidly, the velocity distribution of the hydraulic jump is very much complicated and changes with time. Through the ADV probe, the velocity in the x-direction is obtained at 7 cm above the bed level. Due to fast transient flow, no conclusion can be made about the velocity profile.

4.1 Dam Break with One Obstacle

ADV placed in upstream of the obstacle: With a single obstacle and ADV placed in front of the obstacle, velocity measurements have been observed for different reservoir depths. Velocity measurement is obtained by the ADV probe is shown in Fig. 5. The hydraulic jump was developed by striking water with an obstacle. The hydraulic jump can be recognized as the limit between high velocity upstream of the

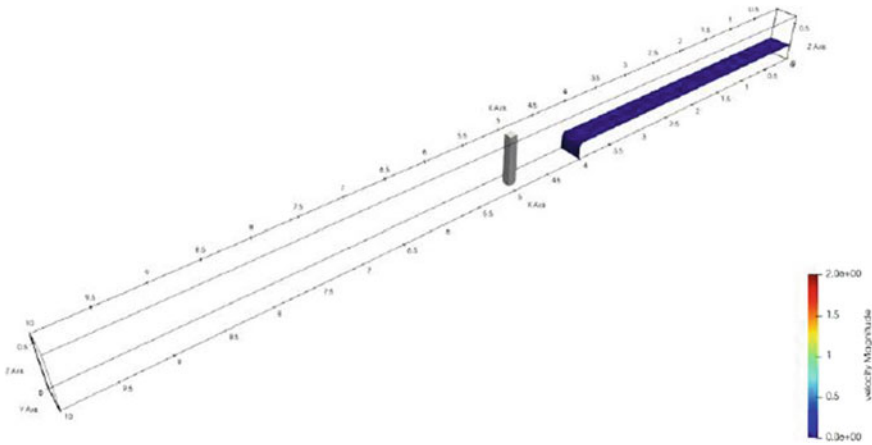


Fig. 4 Geometry of single obstacle model in ParaView

obstacle and a part of water almost at rest. The water separates around the obstacle, and a wake is formed behind this. After some time, the hydraulic jump slowly moves in an upward direction the upstream reservoir empties, the wake zone is as yet present, but the velocities reduce its amplitude.

ADV placed downstream of the obstacle: Figure 6 represents the velocity magnitude in x-direction downstream of the obstacle at different reservoir heights obtained by the ADV.

4.2 Comparison of Velocity Magnitude

ADV placed in upstream of the obstacle: The velocity magnitude results of numerical simulation and velocity magnitude of the experiment is shown in Figs. 7 and 8. Although there is a scatter of data in ADV readings, but a general trend can be obtained.

The velocity magnitude measurement shows the change in velocity from supercritical to subcritical through a hydraulic jump. During the supercritical period, the magnitude of the flow velocity is very high (approximately 2 m/s). Then after the hydraulic jump, the amplitude of the flow velocity decreases rapidly around a value $u = 0.5$ m/s. Finally, the interesting results are obtained at V2, located downstream of the obstacle where the wake zone occurs. The oscillation (periodic change in signs of velocity) proves the wake eddy's presence, which was clearly observed during the experiment.

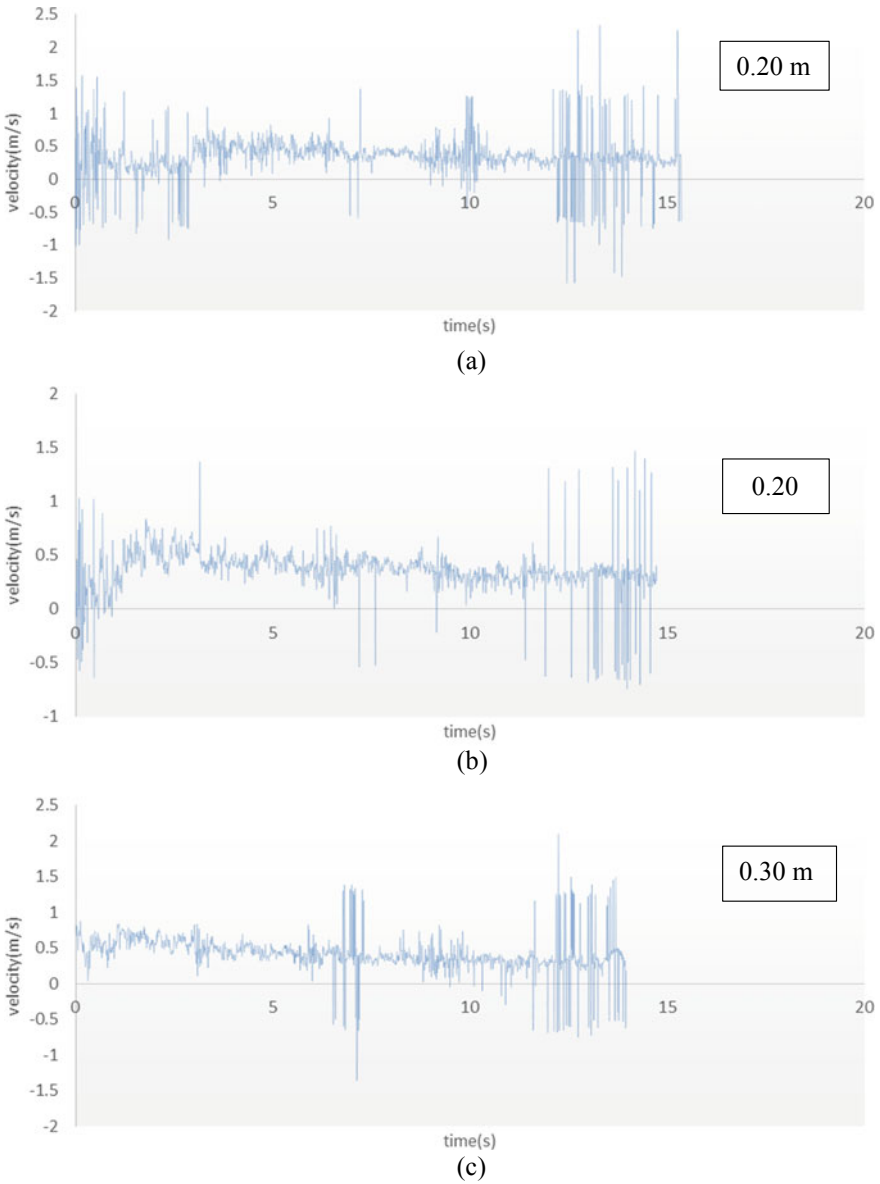


Fig. 5 Velocity magnitude in x-direction upstream of obstacle at different height

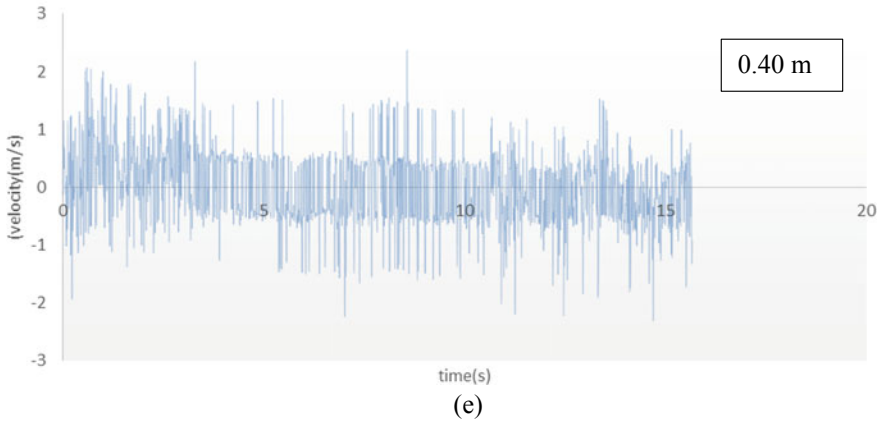
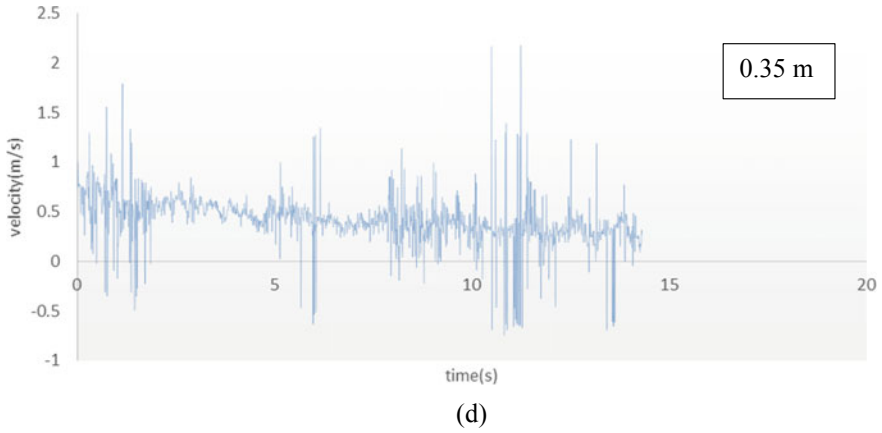


Fig. 5 (continued)

5 Conclusion

In this study, a dam break flow in a flume with a rectangular obstacle is presented. Experimentally and numerically, it is possible to characterize the flow almost completely. In numerical analysis, the level set method is used in REEF3D to visualize the free surface of the fluid, which shows the fluid surface in a very realistic manner. Data measurement from the experimental data set is used to validate the numerical model, aiming to solve complex fast-transient flow problems, which is one of the problem's objectives. The experiments have been performed for different depths upstream. The asymmetrical velocity fluctuation is obtained on the lower depth, and when the depth of the water level is increased, there is the symmetrical velocity fluctuation. After comparing both results, it can be seen that there is an almost similar plot in REEF3D and in experimental work. Therefore, it can be concluded that the

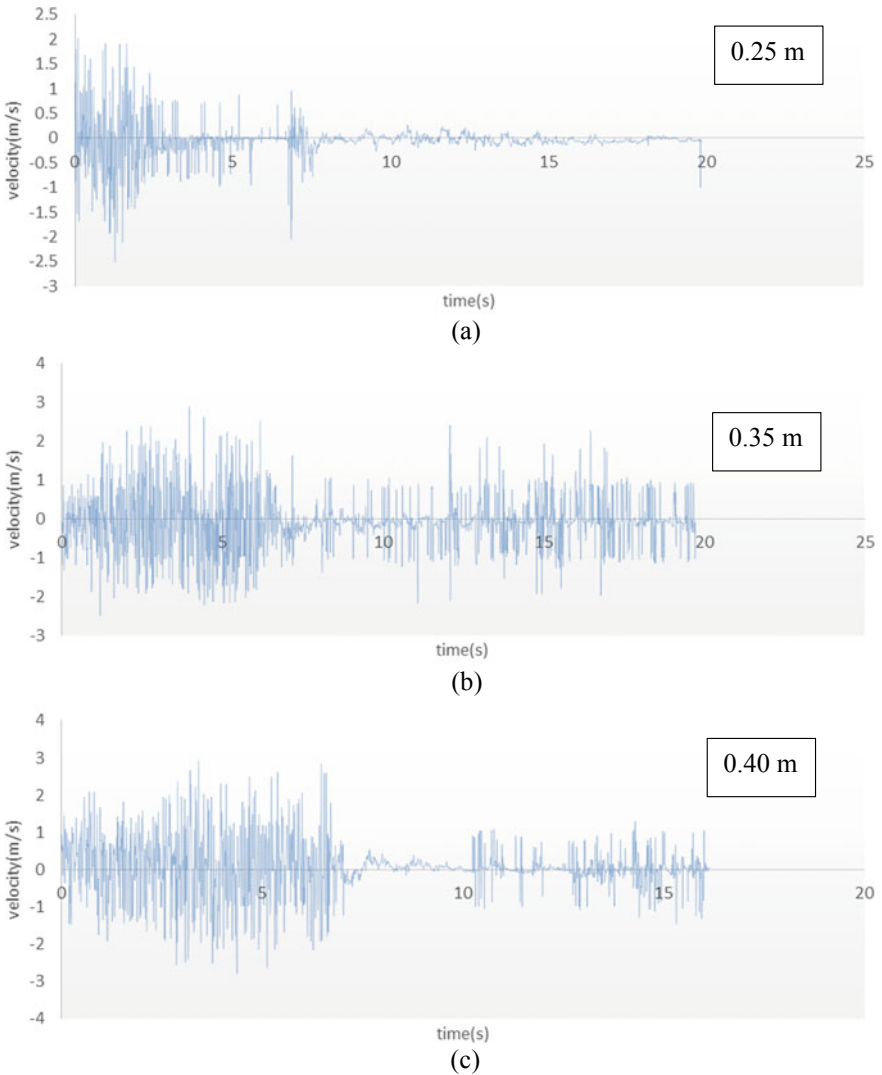


Fig. 6 Velocity magnitude in x-direction downstream of the obstacle at different reservoir height

REEF3D framework can perform the dam break analysis. The present study can be performed with waves and current in future research work. The obstacle shape, size, and position can be changed to determine the velocity fluctuations and ow characteristics. The angles between the two obstacles can be varied to observe the ow hydrodynamics.

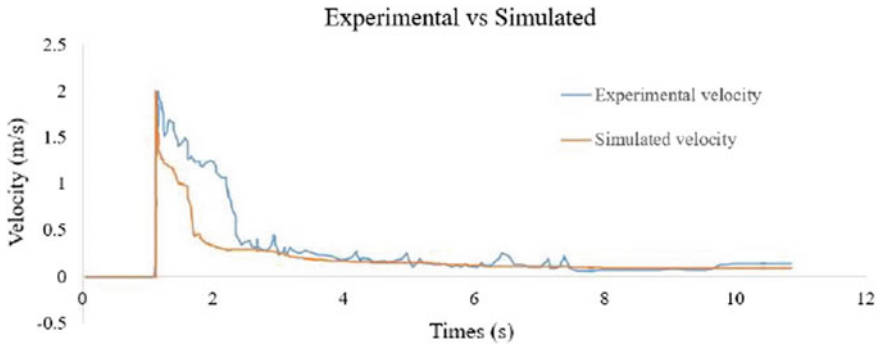


Fig. 7 The comparison of velocity magnitude results obtained in front of obstacle

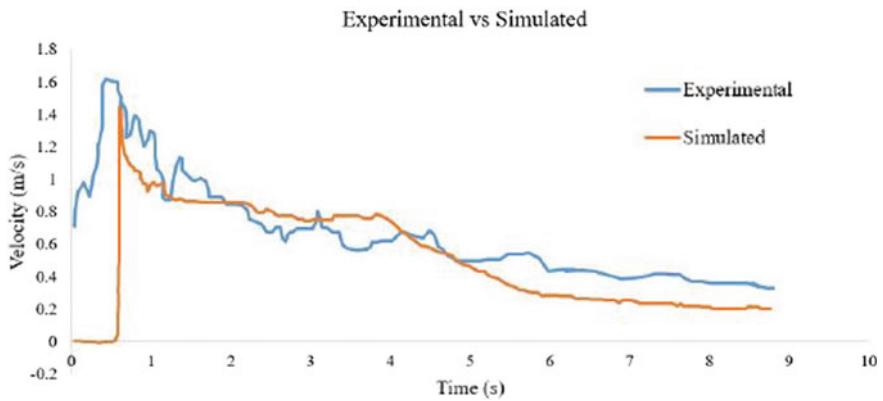


Fig. 8 The comparison of velocity magnitude results obtained after the obstacle

Acknowledgements Authors acknowledge support by SRIC, IIT Kharagpur, under the ISIRD project titled 3D CFD Modelling of the Hydrodynamics and Local Scour Around Off-shore Structures Under Combined Action of Current and Waves.

References

1. Bell SW, Elliot RC, Chaudhry MH (1992) Experimental results of two-dimensional dam-break flows. *J Hydraul Res* 30(2):225–252
2. Leal JG, Ferreira RM, Cardoso AH (2006) Dam-break wave-front celerity. *J Hydraul Eng* 132(1):69–76
3. Dutta D, Mandal A, Afzal MS (2020) Discharge performance of plan view of multi cycle W-form and circular arc labyrinth weir using machine learning. *Flow Meas Instrum* 73:101740
4. Gazi AH, Afzal MS, Dey S (2019) Scour around piers under waves: Current status of research and its future prospect. *Water* 11(11):1–14

5. Kumar L, Afzal MS, Afzal MM (2020) Mapping shoreline change using machine learning: a case study from the eastern Indian coast. *Acta Geophys* 1–17
6. Ritterm A (1892) Die fortanzung der wasserwellen. *Zeitschrift des Vereines Dtsch Ingenieure* 36(33):947–954
7. Hunt JD (1984) Steady state columnar and equiaxed growth of dendrites and eutectic. *Mater Sci Eng* 65(1):75–83
8. Nsom B, Debiane K, Piau J-M (2000) Bed slope effect on the dam break problem. *J Hydraul Res* 38(6):459–464
9. Chanson H (2004) *Hydraulics of open channel flow*, 2nd edn. Elsevier
10. Ancey C, Iverson RM, Rentschler M, Denlinger RP (2008) An exact solution for ideal dam-break floods on steep slopes. *Water Resour Res* 44(1)
11. Liu W, Wang B, Guo Y, Zhang J, Chen Y (2016) Experimental investigation on the effects of bed slope and tailwater on dam-break flows. *J Hydrol* (590):125256
12. Froehlich DC (1989) Proceedings of the 1989 national conference on hydraulic engineering. ASCE, New Orleans, Louisiana, United States
13. Ismail H, Ann Larocque L, Bastianon E, Hanif Chaudhry M, Imran J (2020) Propagation of tributary dam-break flows through a channel junction. *J Hydraul Res* 1–10
14. Aureli F, Maranzoni A, Mignosa P, Ziveri C (2008) Dam-break flows: acquisition of experimental data through an imaging technique and 2D numerical modeling. *J Hydraul Eng* 134(8):1089–1101
15. Soares-Fraza S, Zech Y (2007) Experimental study of dam-break flow against an isolated obstacle. *J Hydraul Res* 45(sup1):27–36
16. Lariyah MS, Vikneswaran M, Hidayah B, Muda ZC, Thiruchelvam S, Abd Isham AK, Rohani H (2013) IOP conference series: earth and environmental science, IOP Conference Series 2013, vol. 16, pp. 012044. <https://doi.org/10.1088/1755-1315/16/1/012044>
17. Feizi A (2018) Hydrodynamic study of the flows caused by dam break around downstream obstacles. *Open Civil Eng J* 12(1)
18. DeMaio A, Savi F, Sclafani L (2004) 3D mathematical simulation of dam-break flow. In: Proceeding of the IASTED international conference environmental modeling and simulation 2004, pp 22–24
19. Bai Y, Dong XU, Lu D (2007) Numerical simulation of two-dimensional dam-break flows in curved channels. *J Hydrodyn Ser B* 19(6):726–735
20. Yi X (2021) A dam break analysis using HEC-RAS. *J Water Resour Prot*
21. Afzal MS, Bihs H, Kumar L (2020) Computational fluid dynamics modeling of abutment scour under steady current using the level set method. *Int J Sedim Res* 35(4):355–364
22. Bihs H, Afzal MS, Kamath A, Arntsen A REEF3D: an advanced wave energy design tool for the simulation of wave hydrodynamics and sediment transport.
23. Ahmad N, Bihs H, Myrhaug D, Kamath A, Arntsen A (2018) Three-dimensional numerical modelling of wave-induced scour around piles in a side-by-side arrangement. *Coast Eng* 138:132–151