Jordan Journal of Mechanical and Industrial Engineering

Numerical Investigation on Dam-Break Flood Wave

Behrouz Aghebatie, Khosrow Hosseini*

Civil Engineering Faculty. Semnan University, Semnan, Iran

Received 4 Jan. 2019

Abstract

The dam-break is one of the pressing issues in rivers. Dam-break causes immense damages and serious threat to human life in the downstream reach. The purpose of this paper is to determine the convenient height of the obstacle and its location to attenuate the flood wave height induced by a dam-break, reduces the serious threats downstream of the failed dam. In this study, the flood wave induced by a dam-break and its evolution is modelled numerically using FLUENT software. The different locations of the rectangular obstacle are investigated to find its influences in time to peak and the height of flood wave. The results showed that the favorable conditions for dealing with flood wave are obtained for the obstacle having 0.66 times the inception wave height and situated 1.1 times of reservoir length in downstream.

© 2019 Jordan Journal of Mechanical and Industrial Engineering. All rights reserved

Keywords: Dam-break; Flood wave; FLUENT; Obstacle; Reservoir;

1. Introduction

A dam is one of the most important hydraulic structures which has the essential role in water resources management. Consequences of dam collapse are enormous, for example, the failure of the Iruka lake dam in Japan, which under the influence of heavy rain this soil dam collapsed and killed more than 940 people. Another example was the collapse of the Puentes dam in Spain which killed more than 600 people and destroyed 1,800 houses and 40,000 trees. Following the identification of the destructive effects of dam-break, the preventative measures must be investigated. The dam-break wave can propagate in a short time, therefore, performance measures to attenuate its height are a required element in the floodplan management in downstream of a dam [1, 2]. The analysis of dam-break has been the subject of the most researches in the last century. Nowadays, by developments achieved in computer sciences, the numerical models offer more accurate results by saving money and time compared with the physical models. One of the first studies regarding the dam-break problem was done by Martin and Moyce [3] that analyzed kinematics of the dam-break wave. Their studies showed that the wave front velocity was proportional to the root of original column height. Further, Dressler [4] investigated the free surface profile and the effect of bed friction in the dam-break over an initially dry bed and confirmed the theoretical solution of the early stages of the dam-break. Nsom et al. [5] have investigated experimentally the effect of bed slope on the dam-break problem. The positive and negative wave fronts can be determined based on the characteristic equations. Also, the drying front computed, and the effect of viscosity can be analysed. They showed that the discharge as the product of depth times velocity demonstrated to be nearly independent of the bottom slope.

The formerly numerical analysing of the propagation of flood wave is accomplished

applying an approach of shallow water equations (SWE), which prepare a proper forecast of unsteady regions of water rising before the obstacle. But, the SWE method is not able to provide information such as negative wave advancing, which arises when a flood wave is fully or partly returned by an obstacle and fails to anticipate exactly pressure distribution over the walls of the obstacle. Also, the SWE approach does not forecast the propagation speed of the flood wave and reflected waves accurately and does not present details of their free surface profile [6,7,8].

Cagatay and Kocaman [9] solved the Reynolds Averaged Navier Stokes equations with the Shallow Water Equations and the k- ε turbulence model. The results indicated that the situation of an obstacle causes reflection of the flood wave and formation of a negative bore which propagates in the upstream of the obstacle.

Leal et al. [10] studied the influence of wave height and the transport of bed materials in downstream channel induced by dam-break. They showed that the presence of shallow water in downstream channel reduces the maximum wave height in the downstream. The opposite trend is observed for the existence of deep water in the downstream. They have also studied the effect of water depth at peak time. Lobosky et al. [11] compared the experimental measurements of dam-break flow on a dry horizontal bed for two different initial heights in the reservoir. Details on both kinematics and dynamics of flow produced from dam-break are compared with the data

^{*} Corresponding author e-mail: khhoseini@semnan.ac.ir.

reported by the other researchers. A reasonable agreement on free surface evolution has been achieved for the wave height as well as the wave front velocity. Khrabry et al. [12] have studied the interaction of dam-break with a triangular obstacle. They showed that the friction of the bottom wall leads to the formation of separation "bubbles" and the occurrence of associated hills on the free surface, depending on the flow development phase. They founded an analytical model to predict the phenomenon and reported a good agreement between analytical solutions and experimental results. Soarez and Zech [13] modelled the turbulent flow in a channel with a rectangular obstacle, representing a building, placed immediately in downstream of the dam. They observed a violent impact of the wave on the building, changing the direction of flow and the formation of hydraulic jumps with the local growth of water level. In five different locations, the water level evolution and the velocity of the wave were measured.

Yang et al. [14] developed a three-dimensional (3D) numerical model to simulate near-field, dam-break flows and to estimate the impact force on obstacles. In their model, the governing equations were solved by a projection method and the water surface was captured by the volume of fluid method (VOF). They reported that the performance of a 3D-VOF model is slightly better than a 2D model in predicting the sharp increase in water level at the wave front and the configuration of the hydraulic jump. This improvement might be due to increasing of the accuracy in capturing the free surface by using the VOF method. The numerical model also indicated that the pressure distribution in the wavefront is not hydrostatic. Park et al. [15] focused on the effects of this basal resistance on the unsteady motion of dam-break flows numerically. For this purpose, a volume of fluid advection algorithm is coupled with the Reynolds Averaged Navier-Stokes (RANS) equations. A two-equation turbulence closure model was employed to introduce the turbulence effects. In order to predict the degree of turbulence in dambreak phenomenon efficiently, the computations were carried out with the variation of initial turbulence intensities. A satisfactory agreement between the numerical and physical results is observed. Aghebatie and Hosseini [16,17] and, Bazargan and Aghebatie [18] modelled the turbulent flow in a chute, using FLUENT software. They simulated the flow by the volume of fluid method and the standard k-ɛ turbulence model. They investigated the occurrence of roll waves for the discharges up to 30 m³/s, the longitudinal slope varying from 17 to 20% and the width of chute varying from 3.4 to 4.0 m. Their results indicated that the formation of roll waves could be modelled numerically as well as experimentally.

In the present study, wave propagation induced by a dam-break is simulated by three- dimensional models. The characteristics of dam-break wave depend upon the reservoir width, the mechanism of dam-break, the slope of lateral walls and water depth in the reservoir. There is no guideline to confront disasters during dam-break. The purpose of this article is to analyse and present the criteria for the control of the flood wave in the dam-break problem from the engineering point of view.

2. Material and Methods

2.1. Governing equations and their solutions

The governing equations described in the form of differential equations could be solved by Computational Fluid Dynamic (CFD) methods. The most popular scheme used in solving the governing equations is the finite volume method. The volume of fluid method (VOF) is used to track the water surface profile. In this method, the volume fraction for each phase is determined at all control volumes. The sum of the volume fractions for all phases in each control volume is unity. Two phases do not interpenetrate. The variables and properties in any given cell are either purely representative of one phase or a mixture of different phases, depending upon the volume fraction values.

The momentum equation is solved in the whole domain and then divided into the different phases. The first term on the left of this equation describes the local change over time. The second term on the left of this equation is the term "Convection". Also, the first term on the right expresses surface forces. The second term on the right represents the term "Diffusion" and the last term expresses the force of gravity and weight. (Yeoh and Tu, 2010; Houichi et al. 2006; and Celik, 1999).

$$\frac{\partial u_{j}}{\partial x_{j}} = 0 \tag{1}$$

$$\frac{\partial \left(\overline{u_{j}}\overline{u_{i}}\right)}{\partial x_{j}} + \frac{\partial \left(\overline{u_{j}'}\overline{u_{i}'}\right)}{\partial x_{j}} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_{i}} + \nu \frac{\partial^{2}\overline{u_{i}}}{\partial x_{j}\partial x_{i}} \tag{2}$$

Where, ρ is the fluid density, p, V and u are the fluid pressure, velocity vector and finally the velocity components, respectively. Also, in the above equation, the superscript T is the transpose sign. The velocity profile of the turbulent flow is highly influenced by the wall. Therefore, the accurate description of the velocity distribution in the vicinity of the wall is of importance. The non-dimensional parameters U^* and y^+ , described by Eqs. (2) and (3), determine whether the flow in the wall adjacent cells is in the viscous sublayer (Chmielewski and Gieras, 2013).

$$U^{*} = \frac{U}{\sqrt{\frac{\tau_{\omega}}{\rho}}}$$

$$y^{+} = \frac{\left(\frac{\tau_{w}}{\rho}\right)^{1/2} y}{U}$$
(4)

In these equations, U is the average velocity and y is the distance from the wall and τ_w is the wall shear stress. If $y^+\rangle(30-60)$, the boundary layer is turbulent and the logarithmic velocity law defined by Eq. 4, is valid for the region in the vicinity of the wall. In FLUENT software this limit is defined by $y^+\rangle 11.225$.

$$U^* = \frac{1}{\kappa} \ln \left(E y^+ \right) \tag{5}$$

In which, k and E are the Von Karman and wall function constants and their values are

0.42 and 9.81, respectively. For y^+ (11.225, the governing velocity equation for the viscous sub-layer is as follows:

$$U^* = y^+ \tag{6}$$

2.2. Turbulence model

One of the main characteristics of turbulent flow is the fluctuation in velocity fields which cause the mixing of transfer phenomenon. Because of the probabilistic nature of turbulence, low and high frequencies and its smallest scales, the turbulence models are correlated with time-averaged of parameters in numerical models. A two-equation turbulence model, called standard k- ε model is used in the present study. Ever since it was proposed in 1974, its popularity in industrial flow simulations has been explained by its robustness, economy and reasonable accuracy for a wide range of turbulent flows (Launder and Spalding, 1974). Applied equations in the model are as follows (Kositgittiwong et al. 2013; *Chanel and Doering, 2008*):

$$\frac{\partial \overline{u_{i}u_{j}}}{\partial t} + U_{k} \frac{\partial \overline{u_{i}u_{j}}}{\partial x_{k}} = -\overline{u_{i}u_{k}} \frac{\partial U_{j}}{\partial x_{k}}$$

$$-\overline{u_{j}u_{k}} \frac{\partial U_{i}}{\partial x_{k}} - \varepsilon_{ij} - M_{ij} - N_{ij}$$

$$\varepsilon_{ij} = 2v \frac{\partial \overline{u_{i}}}{\partial x_{k}} \frac{\partial \overline{u_{j}}}{\partial x_{k}}$$
(8)

$$M_{ij} \equiv \frac{\partial}{\partial \mathbf{x}_{k}} \left(v \frac{\partial}{\partial_{k}} \overline{u_{i} u_{j}} - \overline{u_{i} u_{j} u_{k}} - \frac{\overline{u_{i} p}}{\rho} \delta_{jk} - \frac{\overline{u_{j} p}}{\rho} \delta_{ik} \right)$$
⁽⁹⁾

$$N_{ij} \equiv \frac{p}{\rho} \left(\frac{\partial u_i}{\partial x_j} \frac{\partial u_j}{\partial x_i} \right)$$
(10)

Where, μ indicates the dynamic viscosity, ρ is the

fluid density, **p** is the pressure fluctuation, \mathcal{E}_{ij} is the Reynolds-stress dissipation rate tensor, M_{ij} is the Reynolds-stress transport tensor and N_{ij} is the pressure-rate-of-strain tensor. The variables u_i and u_j are velocities in the x_i , x_j and x_k directions, finally x_i , x_j and x_k correspond to x, y and z coordinates.

2.3. Experimental models

Lobosky et al. (2014) conducted the experiments for a dam-break in a dedicated tank. It consists of a prismatic tank that could be divided into two separate parts by a removable gate and a release system with a sliding mechanism. The length of the prismatic tank is equal to 1.61 m and its width is 0.15 m and the height of the tank is equal to 0.6 m that this model is made of polymethyl methacrylate (PMMA). The dam gate is made of 10mm thick PMMA and is located 0.6 m from the lateral side of the tank. This defined the length of the reservoir area and left 1 m of unobstructed horizontal bottom downstream the dam gate. The initial water depth is 0.3 m with an uncertainty in the filling levels of ±0.5 mm. The different variables such as velocity and wave height were measured along with the model. In Fig 2, the present results are compared with another researchers' study.

2.4. Simulation conditions

The geometry of the experimental setup is modelled in Gambit software, using cubic elements that cells size are varied from 0.008 for areas near the bottom, up to 0.03 for areas without water. The convergence criterion in simulation is satisfied when the residuals of velocity and volume of fluid are inferior to 10⁻⁶ between two consequent iterations. The interface of air-water is determined by evaluating the relative volume of fluid. FLUENT software gives a distinct water surface by linear interpolation. In order to track the interface, the georeconstruction algorithm is used to the discretization of volume fraction. The algorithm can accurately locate the position with interpolation. To solve the governing equations including continuity and momentum equations, segregated solver for multi-phase flow simulation is accomplished. In this study, the first phase for water and the second phase for air are considered. Water is defined as a compressible fluid and air is an incompressible fluid. In the non-compressible flow, the changes of pressure due to changes of velocity are small enough that the small changes of density do not alter the streamlines. The volume of fluid (VOF) model and the standard wall function model are utilized. The turbulence model used in the present study is the standard k-& model. Boundary conditions are defined as the wall for the bottom and lateral walls, radiator for the gate and pressure outlet for the upper space of the models. The radiator boundary type allows us to specify both the pressure drop and heat transfer coefficient as functions of the velocity normal to the radiator. For walls, the non-slip condition is defined. This is the default for all walls in viscous flows. Fluid flows over rough surfaces are encountered in diverse situations. Wall roughness affects drag (resistance) and heat and mass transfer on the walls. To include the Manning roughness coefficient effect in the numerical model, a roughness height of 0.15mm is considered.

$$Q = \frac{AR^{2/3}\sqrt{5}}{n} \tag{11}$$

Where, Q, V, A, n, R and S are the flow rate, velocity, flow area, Manning's roughness coefficient, hydraulic radius and channel slope, respectively. The boundary condition is defined as pressure outlet at the model outlet. In this boundary condition the flow velocity value is then computed based on the fluxes, while the velocity gradient is fixed to zero. A schematic view of the grid mesh and the initial setup of the dam-break simulation in the numerical models are shown in Fig.1.



Figure 1. The grid mesh and the initial setup of the dam-break simulation in the numerical models.

3. Verification tests

The experimental Lobosky's setup (2014) is simulated numerically by a two-phase model. Numerical simulations with various mesh sizes are achieved and the most appropriate mesh size is selected. The criterion proposed by *Kawai and Larsson* (2012) for the validation of logarithmic velocity distribution (y^+ varying from 30 to 300) is also satisfied in selecting the mesh size.

Comparison between experimental results and numerical simulation in the non-dimensional form is also shown in Fig.2.



Figure 2. shows comparison of the numerical and experimental results water front as a function of the relative time.

In this figure, H, t and g are the initial height of the water behind the gate, time after removing the gate and gravity acceleration, respectively. The experimental measurements performed by Martin and Moyce (1952) and Dressler (1954) and Lobosky et al. (2014) are also presented in this figure. As shown in this figure, the form of frontal wave is independent of the initial height in the reservoir. The parameter of RMSE is described as follow:

$$RMSE = \sqrt{\frac{1}{n} \sum_{n=1}^{n} (X_n - X_e)^2}$$
(12)

In which, RMSE is the Root mean squared error, X_n is the numerical value, X_e is the experimental value and n is *number* of *data* samples. Value of RMSE for the numerical model equal to 0.092, is obtained. Therefore, a good adjustment between numerical and experimental data is achieved

4. Results and Discussion

The propagation of flood wave is influenced by different factors such as; channel geometry, waterway roughness and the obstacles located in the channel. Usually, a regulate dam is built downstream of each large dam to regulate the flow released from the main dam for different purposes. The effect of this structure as an obstacle can affect the behaviour of flood wave.

In this study, the regulate dam is modelled by an obstacle, its width varying from 0.1 to 0.2 m and its height varying from 0.05 to 0.1 m, and the location of the obstacle varied from 0.3 to 0.7 m downstream of the gate.

In determining the layout of regulated dams, geological, hydraulic and structural considerations are evaluated. As the regulated dam has the great influences on the wave propagation induced by dam-break, different locations of a rectangular obstacle are investigated to find its influences in time to peak and the height of the flood wave. The numerical simulations are performed for the reservoirs 0.1, 0.15 and 0.2 m in width and 0.15, 0.2, 0.25 and 0.3 m in water depth

Aghebatie and Hosseini studied (2019) slug wave using VOF model along a conduit. Therefore, this reference is used for determining wave height in a channel. In Fig. 3, is shown wave height and its location in the instant of dambreak.



Figure 3. wave height and its location in the instant of dam-break.

The propagation of flood wave is simulated by the FLUENT software with appropriate turbulence model and adequate mesh size achieved in the former section. Graphical visualization facilities in FLUENT help in better understanding the phenomena. In Fig.4, the propagation of the dam-break wave in downstream for two obstacle height and without obstacle is presented.



Figure 4. the flood wave induced by a dam-break a, b and c are without obstacle, 0.2 H and 0.66 H, respectively.

In this figure, two phase model, containing water and air (identified by a colour spectrum varying from red to blue) is shown. This figure inspires the influence of obstacle on the propagation of the dam-break flood wave.

The different heights and locations of the obstacle are investigated by the numerical models. For a reservoir with 0.3 m in length, 0.2 m in width and 0.3 m in water depth, the best configuration of dimensions and location of the obstacle are satisfied by the following conditions. The dam-break wave is completely damped between the reservoir and the obstacle by satisfying the following conditions.

$$Y \ge 0.66h \to Y \ge 0.195 \tag{13}$$

$$l \ge 1.1 \times L$$
 (14)

Where, Y, h, l and L are the obstacle height, wave height in the instant of dam-break, obstacle location and reservoir length, respectively.

In table 1, the relative wave heights induced by dambreak in the downstream channel as a function of reservoir height, obstacle height and its position for different times after the dam-break. As shown in this table, the increase in the obstacle height and its distance from the reservoir, cause the reduction in maximum wave height in the channel.

The relative break-dam wave heights as a function of obstacle height and its location at different times after the occurrence of dam-break are shown in table 2. The maximum wave height is also determined at each point are also shown in this table. It can be concluded from this table that the relative negative wave height downstream of the obstacle is reduced and the relative positive wave height upstream of the obstacle is increased by increasing the obstacle height.

For a rectangular reservoir with the channel width of 0.15 m and the water depth of 0.3 m, during the flood propagation, the minimum and maximum wave height were obtained 0.033 and 0.135 m, respectively. In the other numerical model, similar results were obtained, and the flood wave height varied in the range of 0.25 to 0.78 times of the water depth in the reservoir.

Table 1. Relative wave height	(h/H) in downstream channe	l corresponding its occurrence time.
-------------------------------	------	------------------------	--------------------------------------

Reservoir and Obstacle Dimensions	Time	Distance (cm)						
		0.6	0.8	1	1.2	1.4	1.6	1.8
H=0.15m b=0.20m Y=0.66 H l=0.67 L	t=0.22	0.64	0.053	0	0	0	0	0
	t=0.77	0.50	0.48	0.94	0	0	0	0
	t=1.1	0.486	0.606	0.846	0	0	0	0
	t=1.51	0.586	0.64	0.686	0	0	0	0
	Max	0.64	0.64	0.846	0	0	0	0
H=0.20m b=0.20m Y=0.4H l=0.85 L	t=0.22	0.475	0.04	0	0	0	0	0
	t=0.73	0.375	0.31	0.24	0.175	0	0	0
	t=1.1	0.295	0.29	0.415	0.69	0	0	0
	t=1.72	0.22	0.36	0.455	0.505	0	0	0
	Max	0.475	0.36	0.455	0.69	0	0	0

 Table 2. Relative waves height (h/H) versus the relative obstacle height situated in downstream of a reservoir with 0.3 m in water depth, 0.2 m in width and 0.3 m in length.

Numerical models	Time	Point length (cm)							
		0.6	0.8	1	1.2	1.28	1.4	1.6	1.8
	t=0.22	0.68	0.16	0	0	0	0	0	0
	t=0.71	0.44	0.383	0.32	0.246	0.22	0.186	0.126	0.06
H=0.3 b=0.1 Y=0	t=0.94	0.35	0.326	0.31	0.27	0.253	0.230	0.19	0.156
	t=1.68	0.19	0.190	0.193	0.20	0.206	0.216	0.276	0.63
	Max	0.68	0.383	0.32	0.27	0.253	0.230	0.276	0.63
	t=0.21	0.69	0.146	0	0	0	0	0	0
H=0.3	t=0.71	0.436	0.416	0.34	0.403	0.123	0.516	0	0
b=0.1	t=0.94	0.35	0.336	0.346	0.49	0.57	0.486	0.15	0
Y=0.2 H l=1.1 L	t=1.83	0.236	0.32	0.35	0.356	0.316	0.203	0.263	0.343
	Max	0.69	0.416	0.35	0.49	0.57	0.516	0.263	0.343
H=0.3 b=0.1 Y=0.4 H l=1.1 L	t=0.24	0.64	0.226	0	0	0	0	0	0
	t=0.72	0.43	0.386	0.356	0.586	0.873	0	0	0
	t=0.94	0.31	0.343	0.473	0.703	0.636	0.37	0	0
	t=1.68	0.403	0.42	0.436	0.456	0.416	0	0.006	0.05
	Max	0.64	0.386	0.473	0.703	0.873	0.37	0.006	0.05
	t=0.23	0.68	0.173	0	0	0	0	$\begin{array}{c ccccccccccccccccccccccccccccccccccc$	0
H=0.3 b=0.1 Y=0.6 H l=1.1 L	t=0.71	0.436	0.39	0.36	0.716	0	0	0	0
	t=0.95	0.35	0.36	0.443	0.87	0.816	0	0	0
	t=1.68	0.44	0.453	0.48	0.486	0	0	0	0
	Max	0.268	0.453	0.48	0.87	0.816	0	0	0
	t=0.23	0.68	0.173	0	0	0	0	0	0
H=0.3 b=0.1 Y=0.66 H	t=0.71	0.436	0.39	0.36	0.716	0	0	0	0
	t=1.04	0.32	0.393	0.55	0.803	0	0	0	0
	t=1.68	0.446	0.463	0.473	0.48	0	0	0	0
l=1.1 L	Max	0.68	0.463	0.55	0.803	0	0	0	0

All numerical model studies revealed that in the occurrence of dam-break, the obstacle which could have the convenient influences on the reduction of damages must satisfy the following conditions; the height of the obstacle is 0.66 times the reservoir height and the location of the obstacle is 1.1 times the reservoir length in the downstream reach.

$$Y = 0.66 \times h \tag{15}$$
$$l = 1.1 \times L \tag{16}$$

In Fig. 5, the variation of dynamic pressure along the channel for the instant of collision of dam-break wave to the obstacle is shown. As can be seen in this figure, the maximum dynamic pressure computed equal to 70 Pa and this value determined about 23 Pa at the location of the gate.



Figure 5. Comparison between various distances to measure the dynamic pressure of waves for case b.

Also, results show that the maximum dynamic pressure is 3 times of the dynamic pressure at the location of the gate.

$$P_{\max} = 3 \times p_{gate} \tag{17}$$

Where, P_{max} is the maximum dynamic pressure and p_{gate} is the dynamic pressure at the location of the gate.

In this research, the criteria for the design of obstacles and the measures to decrease the damages in the downstream channel by optimizing the height and the location of obstacles are presented. Comparison between hydraulic characteristics of current including velocity and flow depth shows that the width of the reservoir has important effects on the flow depth, propagation velocities of the wave and peak discharge due to dam-break. Relying on the results of numerical models, by decreasing the width of the reservoir, the water surface of along the reservoir is increased.

5. Conclusions

This research shows that the numerical model can be utilized as a design tool for determining the depth of flow, velocity and the dynamic pressure of the propagated wave induced by a dam-break. In the numerical models, the volume of fluid method and k- ε turbulence model are utilized. A good agreement is observed between experimental and computed results. After the verification tests, the numerical simulations for different conditions such as: different reservoir height, different reservoir width and different height of obstacle are performed. The following results are obtained:

- The height of flood waves varied in the range of 0.25 to 0.78 times of water depth in the reservoir. The obstacle height and its location have the greatest influences on the height of flood wave height which can be introduced as the design criteria for regulated dams. $0.25H \le h \le 0.78H$ (18)
- The maximum dynamic pressure induced by the dambreak wave is equal to 3 times the dynamic pressure at the location of the gate.
- Results showed with reducing the reservoir width, the value of wave velocity is increased which causes the dam-break wave to reach its maximum height in quick time.
- The best configuration of dimensions and situations are obtained for the obstacle having 0.66 times the inception wave height and situated 1.1 times of reservoir length in downstream.

Reference

- A. J. Aisenbrey, R. B. Hayes, H. J. Warren, D. L. Winsett, and R. B. Young, Design of Small Channel Structures. Bureau of Reclamation: Denver; 1978.
- [2] H. Chanson, Embankment Overflow Protection Systems and Earth Dam Spillways. Nova science publishers; New York, 2009.
- [3] J. C. Martin, W. J. Moyce, and Part IV. "An experimental study of the collapse of liquid columns on a rigid horizontal plane," Philosophical Transactions of the Royal Society A, Vol. 244 No. 8, pp.312-324.
- [4] R. F. Dressler, "Comparison of theories and experiments for the hydraulic dam-break wave," International Association of Hydrological Sciences, Vol. 38 No. 3, pp.319-328.
- [5] B. Nsom, K. Debiane, and J. M. Piau"Bed slope effect on the dam-breakproblem,"
- Journal of Hydraulic Research, Vol. 38 No. 6, pp.459-464.
- [6] D. Liang, R. A Falconer, and B. Lin. "Comparison between TVD-MacCormack and ADI-type solvers of the shallow water equations," Advances in Water Resources, Vol. 29 No. 12, pp.1833-1845.
- [7] Q. Liang, and A. G. L. Borthwick. "Adaptive quadtree simulation of shallow flows with wet-dry fronts over complex topography," Computers & Fluids, Vol. 38 No. 2, pp.221-234.
- [8] J. Singh, M. S Altinakar, M. S. and Y. Ding, "Twodimensional numerical modeling of dam-break flows over natural terrain using a central explicit scheme," Advances in Water Resources, Vol. 34 No. 10, pp.1366-1375.
- [9] H. Q. Cagatay, and S. Kocaman. "Dam-break flow in the presence of obstacle: experiment and CFD simulation," Engineering Applications of Computational Fluid Mechanics, Vol. 5 No. 4, pp.541-552.
- [10] J. Leal, R. Ferreira, and A. Cardoso, "Maximum level and time to peak of dam-break waves on mobile horizontal bed," Journal of Hydraulic Engineering, Vol. 135 No. 11, pp.995-999.
- [11] L. Lobosky, E. Botia-Vera, F. Castellana, J. Mas-Soler, and A. Souto-Iglesias, "Experimental investigation of dynamic pressure loads during dam-break," Journal of Fluids and Structures, Vol. 48 No. 1, pp.407-434.
- [12] A. Khrabry, E. Smirnov, D. Zaytsev, and V. Goryaghev, "Numerical study of 2D and 3D separation phenomena in the dam-break flow interacting with a triangular obstacle," Periodica Polytechnica Civil Engineering, Vol. 60 No.3, pp.159-166.

- [13] F. S. Soares, and Y. Zech, "Experimental study of dam-break flow against an isolated obstacle," Journal of Hydraulic Research, Vol. 45 No. 1, pp.27-63.
- [14] C. Yang, B. Lin, C. Jiang, and Y. Liu, "Predicting near-field dam-break flow and impact force using a 3D model," Journal of Hydraulic Research, Vol. 48 No. 6, pp.784-792.
- [15] R. Park, K. S. Kim, J. Kim, and S. H. Van, "Numerical investigation of the effects of turbulence intensity on dambreak flows," Ocean Engineering, Vol. 42 No. 1, pp.176-187.
- [16] B. Aghebatie, and K. Hosseini, "Investigation on the formation of roll waves in chutes," Water and Environment Journal, Vol. 30 No. 1-2, pp.113-118.
- [17] B. Aghebatie, and K. Hosseini, "Analyzing the turbulent flow on steep open channels," Water Science and Technology: Water Supply, Vol. 16 No. 5, pp.1207-1213.
- [18] J. Bazargan, and B. Aghebatie, "Numerical analysis of roll waves in chutes," Water Science and Technology: Water Supply, Vol. 15 No. 3, pp.517-524.
- [19] G. H. Yeoh, and J. Tu, Computational Techniques for Multiphase Flows. Elsevier Ltd: Missouri; 2010.
- [20] L. Houichi, G. Ibrahim, and B. Achour, "Experiments for the discharge capacity of the siphon spillway having the creagerofitserov profile," International Journal of Fluid Mechanics Research, Vol. 33 No. 5, pp.395-406.

- [21] FLUENT Inc. FLUENT 6.2 user guide. http://www.engres. odu. edu/ Applications/ FLUENT 6.2. 2004.
- [22] I. B. Celik, Introductory turbulence modelling. Lecture notes .Western Virginia University: Morganatown; 1999.
- [23] M. Chmielewski, and M, Gieras, "Three-zonal wall function for k-ε turbulence models," Computational Methods in Science and Technology, Vol. 19 No. 2, pp.107-114.
- [24] B. E. Launder, and D. B. Spalding, The numerical computation of turbulent flows, Computer. Methods in Applied Mechanics and Engineering, Vol. 3 No. 2, pp.269-289.
- [25] D. Kositgittiwong, C. Chinnarasri, and P. Y. Julien, "Numerical simulation of flow velocity profiles along a stepped spillway," Proceedings of the Institution of Mechanical Engineers, Part E, Vol. 227 No. 4, pp.327-335.
- [26] P. G. Chanel, and J. C. Doering, "Assessment of spillway modelling using computational fluid dynamics," Canadian Journal of Civil Engineering, Vol. 35 No.12, pp.1481-1485.
- [27] S. Kawai, and J. Larsson, "Wall-modelling in large eddy simulation: length scales, grid resolution and accuracy," Physics of Fluids, Vol. 24 No. 1, pp.399-412.
- [28] B. Aghebatie, and KH. Hosseini, (2019), "Computational investigation on the effects of rib on the slug flow phenomenon; using OpenFOAM", International Journal of Modern Physics C. Vol. 30 No. 6 pp. 1-12.