Study of Hydraulic Characteristics of Large Dam's Spillway (Case Study: Zola Dam Spillway)

Mohammad Manafpour[1], Sevda Kafshdooz Salimy[2]

1- Assistant Professor of Civil Engineering Department, Urmia University, Urmia, Iran
2- Ph.D. Student of Civil Engineering- Hydraulic Structures, Urmia University, Urmia, Iran

Email: Sevda.salimy@yahoo.com

Abstract

In the past century, various types of the large dams were built on rivers in order to control the water surface flows and supply water needed for domestic, agriculture, industrial sectors, hydroelectric power plants and also for flood control purposes. One of the important hydraulic structures related to this type dam is spillway which plays a key role to discharge safely the flood flow to the downstream of dam without causing any damage to the dam structure; therefore, in large dam's spillway considering the formation of turbulent flow, assessing the pressure and velocity is important. In order to optimize the geometry of spillways, the study of hydraulic performance and pattern of flow past spillways, longitudinal water surface profile is necessary. In the present research, the pattern and characteristics of flow over Zola Dam spillway as a case study are numerically investigated using FLUENT software k-epsilon RNG turbulent model and multi-phase. The VOF method is applied to simulate the free surface flow. The numerical model was verified using physical model results. Comparison of the results of the numerical model with those of the physical model indicates that the numerical model is able to appropriately predict the air-water flow pattern as well as velocity and pressure fields.

Keywords: Spillway, Hydraulic Behavior, Two Phase Flow, Numerical Model, FLUENT.

1. INTRODUCTION

A study on the flow through the hydraulic structures is usually conducted using physical modeling. Physical modeling is based on the basic fluid mechanic equations. Physical modeling of hydraulic structures means that a scaled laboratory model of the prototype is constructed. This approach is a safe way to analyze the flow through or over the hydraulic structures [1]. The spillway is one of the most important hydraulic structures in the hydropower project to ensure the safety of hydraulic structures during the flood. Spillways is a structure which discharges excess water form maximum water level avoiding damage to the dam and its facilities. So the spillway must be carefully designed to verify the flow characteristics [2]. To better understand the flow characteristic, it is recommended to study the pressure and velocity pattern of the flow.

Study of the flow pattern by means of physical models needs time and expense. In recent years due to the development of advanced techniques and accurate software, it is desirable to reduce the cost via numerical models.

Many researches have studied the flow pattern of different types of spillways, for example Zhan et al., Kositgittiwong et al. and Xiangju et al. studied the characteristics of flow over stepped spillways by numerical models [3, 4, 5]. Dargahi investigated flow field over an over-flow spillway and simulated the flow by a three-dimensional (3D) numerical model [6]. Varjavand studied flow pattern over a side spillway.

There are few articles about flow patterns in the ogee and chute spillway. The main purpose of this study was to justify the methodology adopted in ANSYS FLUENT via comparing water surface elevation in physical and simulated model, and also to assess the pressure and velocity in two phase air-water flow.

2. METHODOLOGY

Computational fluid dynamic (CFD) is an advanced numerical approach used along the physical modeling for modeling the hydraulic phenomena. Advances in high-performance computations and the development of computational fluid dynamics general-purpose software have made it possible to investigate the physical reliability of simulations of complex flows measured in reduced-scale models and prototype spillways [7]. The geometry of the spillway was built based on the original layout provided by Iran Water Research Institute. An appropriate meshing will be inserted to match the geometry and the simulation will be initialized with experimental data. Accuracy and validation of the simulations will be carried out using the flow data received from physical model by Iran Water Research Institute.

2.1. CASE STUDY

Zola Dam located at 17 kilometers to Salmas in West Azarbayjan province. It is built on Zola Chay and supplies water needed for domestic, agriculture, industrial sectors, hydroelectric power plants and also for flood control purposes. Zola water basin area is about 945 square kilometers. It is an earthfill reservoir dam and its spillway includes an ogee and a chute. Characteristics of Zola dam are shown in Table 1.

Table 1 : Introduction to Zola Dam

Dam	River	Type	Latitude	Longitude	Width of ogee(m)	Effective storage volume (MCM)	Length of spillway (m)
Zola Dam	Zola Chay	Earthfill with clay core	59.99 '05 38°	00.01 '39 44°	40	72	225.2

2.2. PHYSICAL MODEL

Zola Hydraulic model of spillway is constructed in 1:25 scale. Most parts of the spillway including ogee, chute and stilling basin are made by Plexiglas. Guidance walls are made of wood. There are 183 piezometers in the bottom of the ogee, chute and stilling basin in three axes (center and side walls) to measure the pressure. Also 32 piezometers are located in the side walls of stilling basin to measure the static and dynamic pressure. Physical model of Zola dam is constructed by Iran Water Research Institute and is shown in Figure 1.

Figure 1: Physical Model Of Zola Dam

2.3. DESCRIPTION OF CFD MODEL

Geometric model is simulated by Gambit according to the data collected from West Azarbayjan Regional Water Authority, then the model was exported to Ansys Fluent 16.0 to model the hydraulic flow over the spillway.

In the present study, the diffusion terms are discretized using a second-order accurate central scheme, while the convective terms are discretized using the second-order upwind scheme. The momentum equation is discritisized with second order upwind method. Through turbulence closure models available in Fluent, the k-ε model was chosen because it was found to be the most robust model for different initial conditions [9].

The VOF model was chosen to model the free-surface water-air interface boundary. The VOF is a surface-capturing algorithm in which the interface between water and air is determined by solving an additional water volume fraction transport equation [10]. The model's upstream boundary was set on approach channel in relatively straight channel reaches about 40 m from the spillway ogee to ensure that the flow was not influenced by increasing the bed level. Similarly, the outlet boundary section was located at stilling basin to reduce the effect of omitting the rest of spillway.

The no-slip boundary condition is used. The type of inlet is set on Velocity Inlet boundary condition. The free surface boundary condition is specified as pressure inlet condition. The outlet condition is the pressure outlet condition, with the pressure set equal to atmospheric pressure.

Depending on the position of the cell, Quad meshes were used. The size of 0.15 m was used for the entire ogee and Pave meshes with different sizes (according to the turbulence of flow) in the rest of the spillway, approach channel and stilling basin. The calculation domain and boundary conditions are shown in

Figure 2.

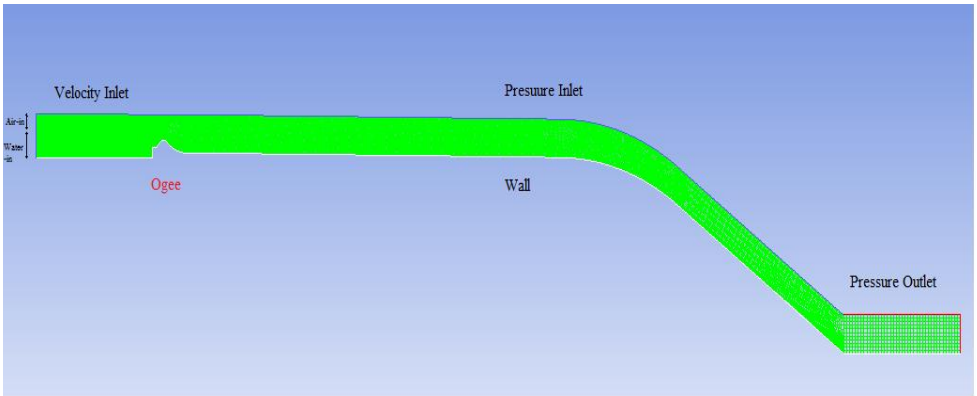

Figure 2: The calculation domain and boundary conditions.

3. VERIFICATION OF NUMERICAL MODEL

The numerical model was verified using the results of the physical model. Comparison of the results obtained of the numerical model with those of the physical model indicates that the numerical model is able to appropriately predict the air-water flow pattern as well as velocity and pressure fields. Depth measurements provided by Iran Water Research Institute using the physical model are applied to validate the accuracy of results. In the case of Q=501.5 m3/s Regression equals to 0.98 and in the case of Q=1006 m3/s Regression equals to 0.95, so the validity of the results is acceptable. Comparison of observed and simulated surface water elevation is shown in Figure 3 and 4. The difference in PMF discharge (Q=1006 m3/s) may have been caused by excessive turbulence.

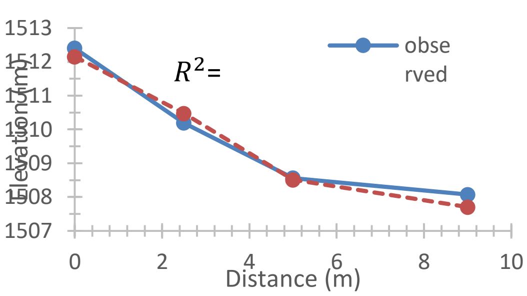

Figure 3: Comparison of observed and simulated surface water elevation for Q=501.5 m3/s

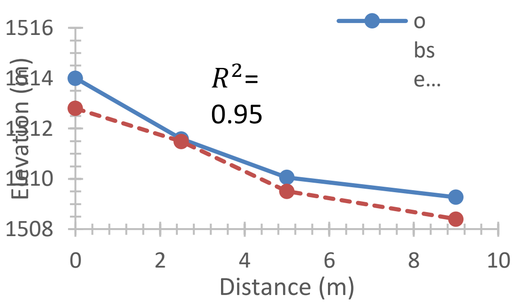

Figure 4: Comparison of observed and simulated surface water elevation for Q=1006 m3/s

4. Simulation Results and Analysis

Pressure and velocity is one of the main characteristics of the flow. To understand the flow pattern and to design safe hydraulic structures, it is necessary to study the pressure and velocity. Here, this goal is achieved by means of Ansys Fluent on the base of CFD. After simulation of geometric model and running the calculation, pressure and velocity are exported in six sections x=0, 2.5, 5, 7, 9 and 10 meters (where (x, y) = (0, 0) shows the ax of ogee) for two discharges Q=501.5 and Q=1006 cubic meter per second which is shown in

Figure 5 and

Figure 6. Also flow pattern is computed as below for two discharges (Figure 7 and 8).

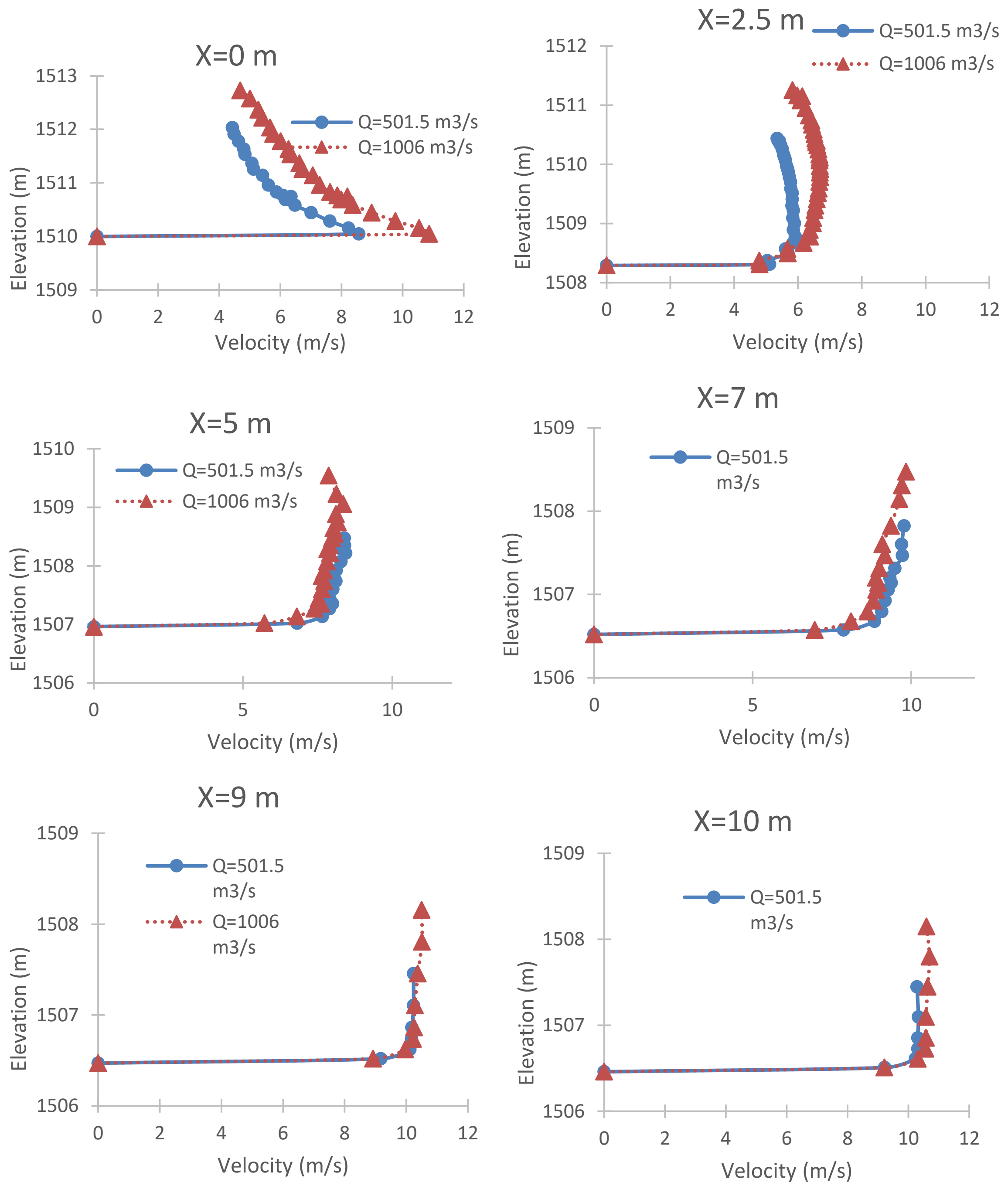

Figure 5:Comparison of velocity profile between discharges of 501.5 m3/s and 1006 m3/s in six sections (x=0, 2.5, 5, 7, 9 10)

According to

Figure 5, at x=0 section, the velocity profile is not fored yet and this may be caused by turbulence of water passing the head of ogee. After X=2.5m, the expected velocity profile is showing off. Comparison of velocity in different sections also clearly reflects the tendency of the flow's velocity to gradually increase.

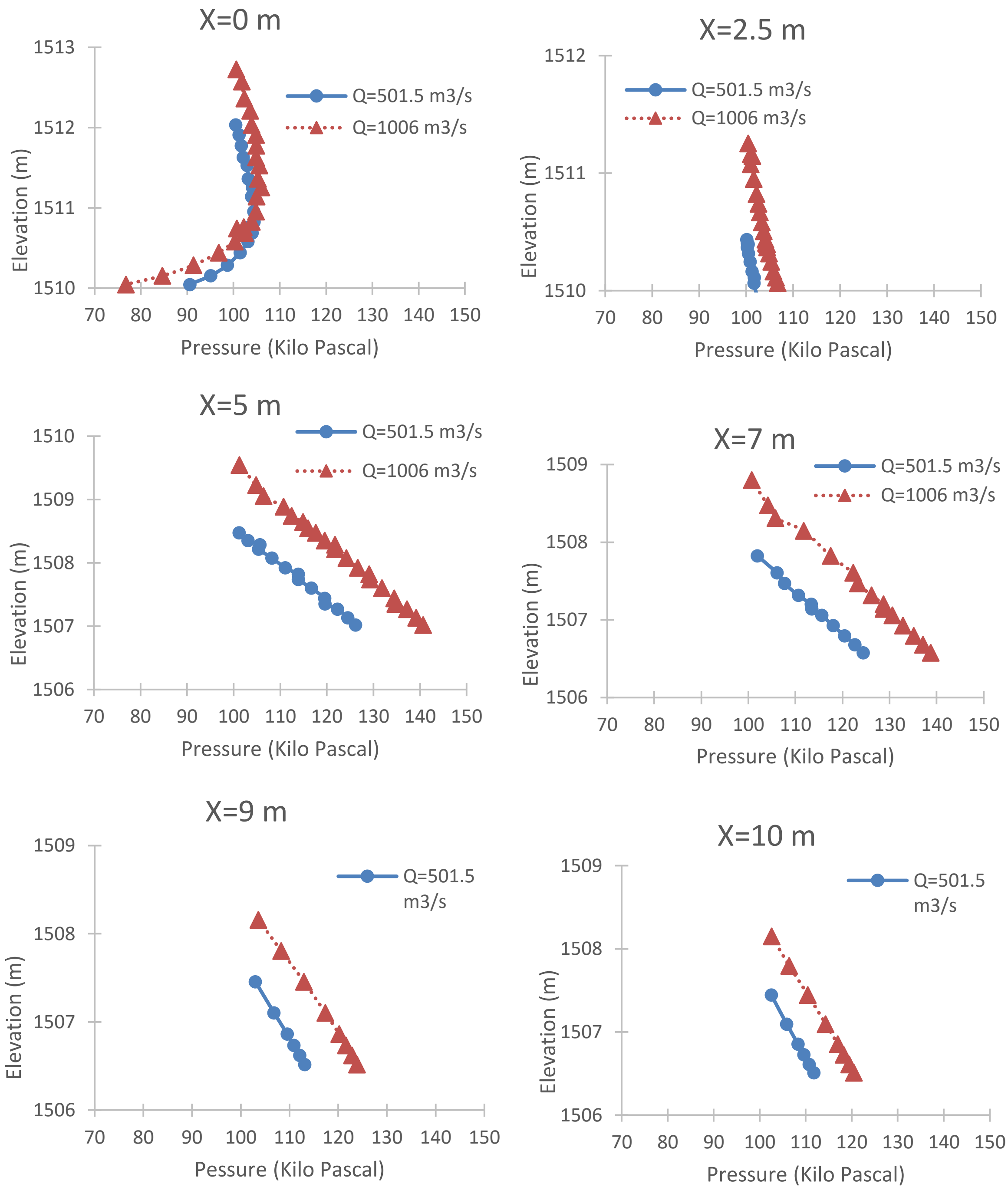

Figure 6: Comparison of static pressure profile between discharges of 501.5 m3/s and 1006 m3/s in six sections (x=0, 2.5, 5, 7, 9 10)

Static pressure is influenced by the hydraulic head of water. According to

Figure 6, it is obvious that the static pressure profile is linear which provides the linear relation between depth of water and static pressure. There is a non-linear pressure profile at section x=0 m which may be caused by turbulence.

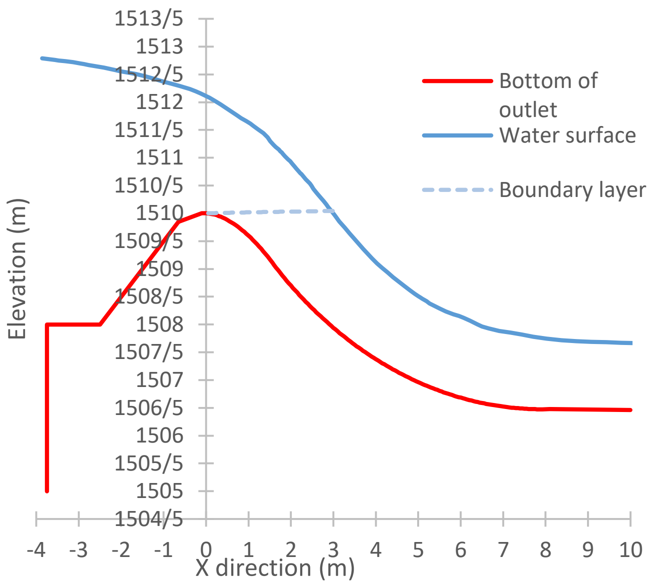

Figure 7:Flow pattern for Q=501.5 m3/s

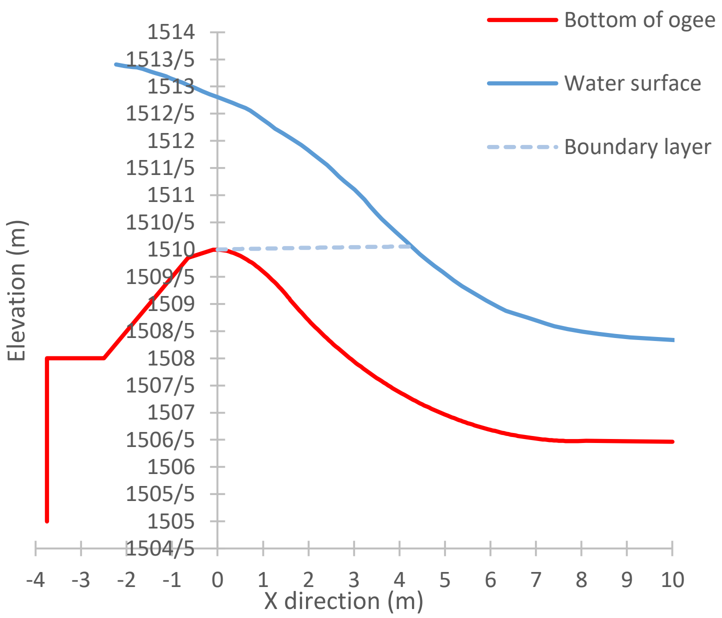

Figure 8: Flow pattern for Q=1006 m3/s

One of the important topics in hydraulic is the distribution of the boundary layer passing through the crest of the ogee. Distribution of boundary layer is highly influenced by the roughness of surface [12]. The process of air entrainment begins where the outer limit of the boundary layer meets the flow surface Boundary layer reaches the surface of flow at x=3m for Q=510.5 m3/s and at x=4.2 m for Q=1006 m3/s which is shown in Figure 7 and Figure 8. This is calculated by Davis Formula which is shown below.

$$\frac{\gamma}{L} = 0.08\left(\frac{L}{k}\right)^{-0.233}$$

Where

γ =	Boundary layer thickness, meter
L=	Surface distance from upstream end of the ogee crest, meter
k =	Absolute concrete-surface roughness (ranges from 0.0006 to 0.002 meter)

5. CONCLUSIONS

A numerical model using multiphase flow, VOF is used to study and compare the flow over an ogee spillway with two different discharges. In the current study, the VOF model for air-water two-phase flow and RNG k-ε turbulence model are combined to successfully simulate the flow characteristics over ogee spillways. The simulated pressure profiles on the ogee surfaces provide the theoretical foundation for assessing the cavitation risk. Moreover, the numerical method in this paper is an essential tool for designing a safe hydraulic structure. The main goal of this study was to justify the methodology adopted herein for CFD model setup, as well as the reliability of κ-ε turbulence closure and VOF method to simulate complex 2D flow fields in open channels and hydraulic structures by means of ANSYS FLUENT. This goal is achieved by negligible difference in comparing the water surface level in the physical and simulated model.

6. REFERENCES

1. Dehdar-Behbahani, S., & Parsaie, A. (2016). Numerical modeling of flow pattern in dam spillway's guide wall. Case study: Balaroud dam, Iran. Alexandria Engineering Journal, 55(1), 467-473.
2. Zhenwei, M. U., Zhiyan, Z., & Tao, Z. H. A. O. (2012). Numerical simulation of 3-D flow field of spillway based on VOF method. Procedia Engineering, 28, 808-812.
3. Zhan, J., Zhang, J., & Gong, Y. (2016). Numerical investigation of air-entrainment in skimming flow over stepped spillways. Theoretical and Applied Mechanics Letters, 6(3), 139-142.
4. Kositgittiwong, D., Chinnarasri, C., Julien, P. Y., Ruff, J. F., & Meroney, R. N. (2010, May). Multiphase Flow Models for 2D Numerical Modeling of Flows over Spillways. In The Third International Junior Researcher and Engineer Workshop on Hydraulic Structures, IJREWHS-10, Edinburgh, Scotland, UK.
5. Cheng, X., Chen, Y., & Luo, L. (2006). Numerical simulation of air-water two-phase flow over stepped spillways. Science in China Series E: Technological Sciences, 49(6), 674-684.
6. Dargahi, B. (2006). Experimental study and 3D numerical simulations for a free-overflow spillway. Journal of Hydraulic Engineering, 132(9), 899-907.
7. Zeng, J., Zhang, L., Ansar, M., Damisse, E., & González-Castro, J. A. (2016). Applications of computational fluid dynamics to flow ratings at prototype spillways and weirs. I: Data generation and validation. Journal of Irrigation and Drainage Engineering, 143(1), 04016072.
8. Varjavand, P., Farsadizade, D., Hosseinzadeh, A., Sadraddini, A. (2009). 3D Simulation of Flow in Side Spillway with k-ε Turbulence Model and Comparing the Results with Physical Model. Journal of water and soil science (in Persian).
9. Wolfgang, R. Shamloo, H. (2009). Turbulence models and their application in hydraulics. Tehran: Khaje Nassir Tousi University (in Persian).
10. Hirt, C. W., and Nichols, B. D. (1981). "*Volume of fluid (VOF) method for the dynamics of free boundaries*." J. Comput. Phys., 39(1), 201–225.
11. Zipparro, V. J., & Hasen, H. (Eds.). (1993). Davis' handbook of applied hydraulics. McGraw-Hill Companies.
12. Manafpour, M., Mirizadeh, N., (2016). Distribution of boundary layers on stepped spillways. Fifteenth Hydraulic Conference of Iran, Imam Khomeini University of Qazvin (in Persian).
13. Davis, C. V., & Sorensen, K. E. (1969). Handbook of applied hydraulics. In Handbook of applied hydraulics. McGraw-Hill.