



Alexandria University  
**Alexandria Engineering Journal**

[www.elsevier.com/locate/aej](http://www.elsevier.com/locate/aej)  
[www.sciencedirect.com](http://www.sciencedirect.com)



ORIGINAL ARTICLE

# Numerical modeling of flow pattern in dam spillway's guide wall. Case study: Balaroud dam, Iran



Sadegh Dehdar-behbahani <sup>a,\*</sup>, Abbas Parsaie <sup>b</sup>

<sup>a</sup> *Shahid Chamran University, Ahvaz, Iran*

<sup>b</sup> *Department of Water Engineering, Lorestan University, Khorramābād, Iran*

Received 2 June 2015; revised 11 January 2016; accepted 19 January 2016  
Available online 3 February 2016

## KEYWORDS

Balaroud dam;  
Cross waves;  
Optimal form of structure;  
Guide wall;  
Approach channel

**Abstract** Flow pattern recognition using the physical modeling and numerical simulation helps designers to propose optimal shape for hydraulics structures. Optimal shape of hydraulic structure causes increasing their performance. In this study, the effect of geometries of guide walls on the flow pattern and rating curve of Balaroud dam spillway's (Iran) was simulated using numerical and physical simulation. Numerical simulation of flow pattern through the guide walls was carried out using Flow 3D software as computational fluid dynamic (CFD) tool. To this task, three plans for the guide walls were evaluated. The results showed that the plan (3) has best performance for removing cross waves and smoothly passing the flow through the approach channel and over the spillway. Evaluating the turbulence models showed that the RNG-K-epsilon becomes the best performance to show the cross waves.

© 2016 Faculty of Engineering, Alexandria University. Production and hosting by Elsevier B.V. This is an open access article under the CC BY-NC-ND license (<http://creativecommons.org/licenses/by-nc-nd/4.0/>).

## 1. Introduction

Study on the flow through the hydraulic structures usually is 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 from the prototype is constructed. This approach is a safe way to analyze the flow through or over the hydraulic structures [9,12,25]. Due to high cost of laboratory experiments, researchers have

attempted to use numerical simulation along the physical modeling. In the field of numerical simulation, the governing equations of the flow which are usually Navier–Stokes equations together with turbulence models are solved with powerful numerical methods such as finite volume and finite elements [17,18,19,20,26]. Several turbulence models have been proposed for solving the turbulent flow. In this regard the K-epsilon, RNG K-epsilon and large eddy simulation (LES) are the advanced turbulence models that recently have been widely used in the hydraulic engineering [4,5,15,22,23,26]. Several commercial software and open source codes have been proposed for comfortable use of the numerical modeling in the hydraulic engineering studies. In this regard, the Flow 3D and Fluent as commercial software and Open-FOAM as free access code can be mentioned [16]. Numerical modeling of turbulent flow

\* Corresponding author.

E-mail addresses: [sadegh.dehdar@gmail.com](mailto:sadegh.dehdar@gmail.com) (S. Dehdar-behbahani), [Abbas\\_Parsaie@yahoo.com](mailto:Abbas_Parsaie@yahoo.com) (A. Parsaie).

Peer review under responsibility of Faculty of Engineering, Alexandria University.

over hydraulic structures helps to define flow properties including velocity and pressure distribution through the structure. In addition to define hydraulic characteristics, numerical models have high ability to show the flow pattern which some time is important as flow properties [21,24]. Defining the flow pattern through the spillway's guide wall is categorized as this subject and guide wall shape has meaningful effect on the flow pattern [1,2,6,7,8,10]. There are few articles about flow pattern in the guide wall but in this regard, Wang and Chen [27,28] studied the spillway of Yutang dam to remove vortex flow and separation on the guide wall. They proposed that the guide wall should be redesigned and present a new shape for the guide wall to elimination remove vortex front the gate. Parsaie et al. [21] numerically studied the effect of geometry of guide walls of Kamal-Saleh (Iran) on the flow pattern passing through the approach channel and over the ogee weir. They stated that the geometry of the right side wall causes cross wave creation through the spillway specifically at the entrance of approach channel. In this paper, physical modeling and numerical simulation of flow through the guide wall and approach channel of the Balaroud dam spillway are considered, whereas the main focus of this paper was on the numerical simulation. To this purpose, the Flow 3D as CFD tool is used. The aim of this paper was finding an optimal shape for the guide wall which removes the cross waves in the maximum flood design. Successfully using the Flow 3D for modeling the hydraulic properties has been reported in the several studies such as souring, waving, cavitation, and energy dissipation [3,11,13]. Using this software for modeling the flow characteristics on the board crest weir was applied by Maghsoodi et al. [14] and they stated that this package has high performance for modeling the flow over this type of weirs.

## 2. Material and methods

Computational fluid dynamic (CFD) is an advanced numerical approach which is used along the physical modeling for modeling the hydraulic phenomena. Using the CFD tools for modeling the flow through the hydraulic structures needs to define 2D or 3D geometry of structure. Because this paper focuses on the flow pattern through approach channel, the 3D computer model of the Balaroud dam spillway was planned in the AutoCAD software (Fig. 1).

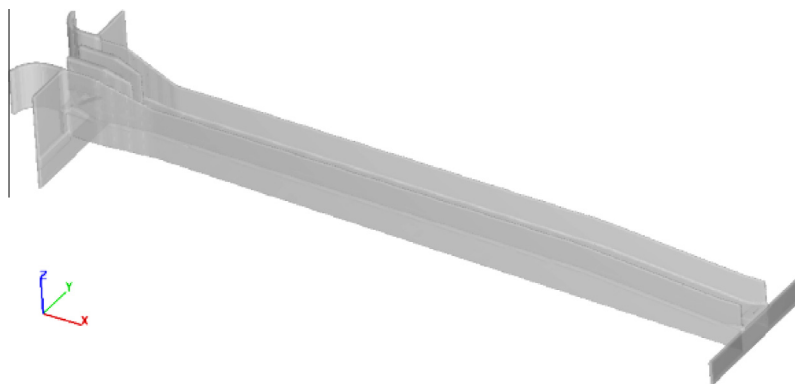


Figure 1 Sketch of 3D model of spillway of Balaroud dam.

The Balaroud dam is an earth dam located at the west south of Iran (Khozestan Province). Fig. 2 shows the scaled laboratory model which was constructed for the study of the flow properties through the spillway. To assess the geometry of guide wall on the flow pattern, three plans were proposed for the guide wall of approach channel. Figs. 3–5 show the computer models and their geometrical properties which were proposed for plans of the guide wall. In this figures the radius of guide wall and length of approach channel are shown. The scale of constructed laboratory model is equal to 1:110 and the plexiglass was used as material for constructing the flume.

CFD modeling by Flow 3D includes few steps which are explained as follows: preparing the 3D computer model by AutoCAD software and exporting the suitable format (stl format) for uploading in the Flow 3D, Summoning the STL file from the Flow 3D, meshing the computational domain, defining the flow characteristics, boundary and initial condition and then running the software. Figs. 6 and 7 show the Balaroud dam spillway in the Flow 3D software. Fig. 6 shows the mashing process and Fig. 7 shows the boundary conditions. In Fig. 6, W shows the wall, S is symmetry and O is outflow boundary conditions. The Discharge of flow is considered for upstream boundary condition.

## 3. Overview on governing equation on the Flow 3D

The continuity equation at three dimensional cartesian coordinates is given as in Eq. (1).

$$v_f \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(uA_x) + \frac{\partial}{\partial x}(vA_y) + \frac{\partial}{\partial x}(wA_z) = \frac{PSOR}{\rho} \quad (1)$$

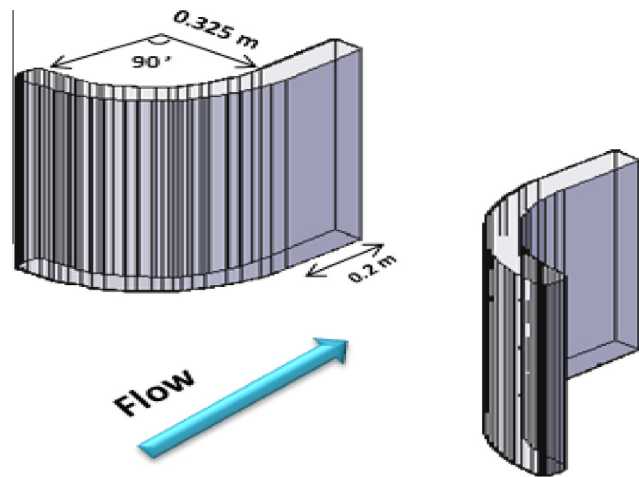
where  $u, v, z$  are velocity component in the  $x, y, z$  direction.  $A_x, A_y, A_z$  are cross-sectional area of the flow,  $\rho$  is the fluid density,  $PSOR$  is the source term,  $v_f$  is the volume fraction of the fluid and three-dimensional momentum equations are given in the Eq. (2).

$$\begin{aligned} \frac{\partial u}{\partial t} + \frac{1}{v_f} \left( uA_x \frac{\partial u}{\partial x} + vA_y \frac{\partial u}{\partial y} + wA_z \frac{\partial u}{\partial z} \right) &= -\frac{1}{\rho} \frac{\partial P}{\partial x} + G_x + f_x \\ \frac{\partial v}{\partial t} + \frac{1}{v_f} \left( uA_x \frac{\partial v}{\partial x} + vA_y \frac{\partial v}{\partial y} + wA_z \frac{\partial v}{\partial z} \right) &= -\frac{1}{\rho} \frac{\partial P}{\partial y} + G_y + f_y \\ \frac{\partial w}{\partial t} + \frac{1}{v_f} \left( uA_x \frac{\partial w}{\partial x} + vA_y \frac{\partial w}{\partial y} + wA_z \frac{\partial w}{\partial z} \right) &= -\frac{1}{\rho} \frac{\partial P}{\partial z} + G_z + f_z \end{aligned} \quad (2)$$

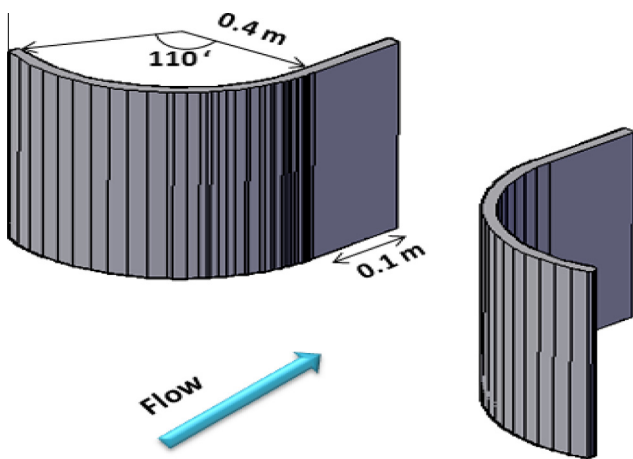
where  $P$  is the fluid pressure,  $G_x, G_y, G_z$  are the acceleration created by body fluids,  $f_x, f_y, f_z$  are viscosity acceleration in



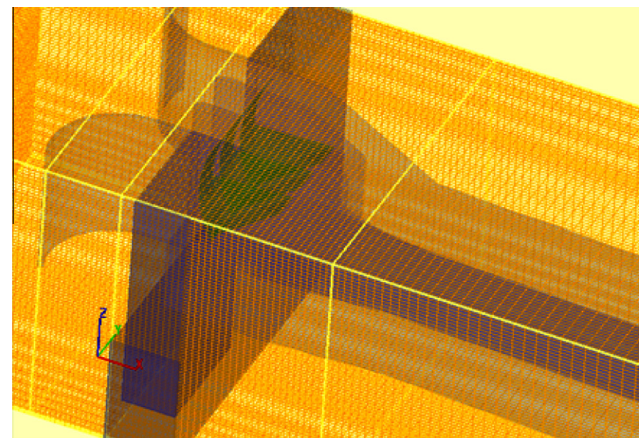
**Figure 2** Sketch of scaled laboratory model of the Balaroud guide wall dam.



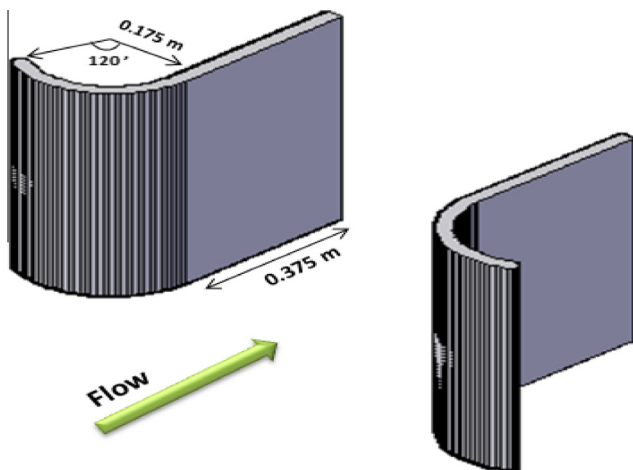
**Figure 5** Model (3) for Balaroud dam guide wall.



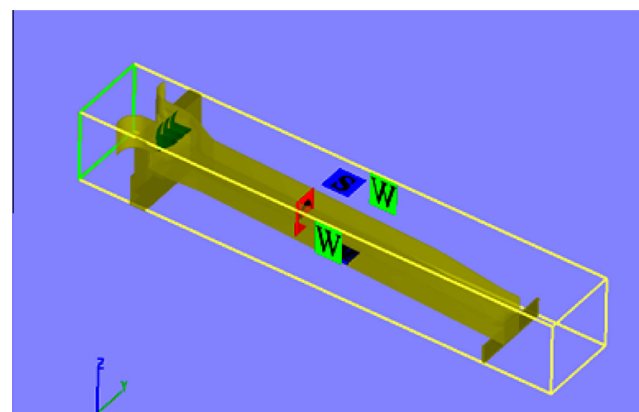
**Figure 3** Model (1) for Balaroud dam guide wall.



**Figure 6** Meshing the Balaroud dam in Flow 3D.



**Figure 4** Model (2) for Balaroud dam guide wall.



**Figure 7** Bindery conditions of Balaroud dam model in Flow 3D.

three dimensions and  $v_f$  is related to the volume of fluid, defined by Eq. (3). For modeling of free surface profile the VOF Technique based on the volume fraction of the computational cells has been used. Since the volume fraction  $F$

represents the amount of fluid in each cell, take value between 0 and 1.

$$\frac{\partial F}{\partial t} + \frac{1}{v_f} \left[ \frac{\partial}{\partial x} (FA_x u) + \frac{\partial}{\partial y} (FA_y v) + \frac{\partial}{\partial z} (FA_z w) \right] = 0 \quad (3)$$

Flow 3D included most popular turbulence models such as K-epsilon, RNG-K-epsilon and large eddy simulation (LES). Modeling turbulent flow requires defining suitable turbulence model, which creates close form with Navier–Stokes equation. Re-Normalization Group (RNG) model is a powerful turbulence model which has suitable performance for modeling the fine vortex; therefore, this model is very useful for modeling flow pattern problems. Most equations used in RNG model are given in Eqs. (4) and (5).

$$\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x} (\rho u_i k) = \frac{\partial}{\partial x_i} \left( \alpha_k \mu_{eff} \frac{\partial k}{\partial x_i} \right) + G_k + G_b - \rho \varepsilon \quad (4)$$

$$\begin{aligned} \frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x} (\rho u_i \varepsilon) &= \frac{\partial}{\partial x_i} \left( \alpha_k \mu_{eff} \frac{\partial \varepsilon}{\partial x_i} \right) \\ &+ C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} - R \end{aligned} \quad (5)$$

in which  $G_k$  is the rate kinetic energy creation, and  $R$  is the density of turbulence defined as below.

$$R = \frac{C_\mu \rho \eta^3 (1 - \eta/\eta_0)}{1 + \beta \eta^3} \frac{\varepsilon^2}{k}, \quad \mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (6)$$

In these equations  $\beta = 0.012$ ,  $\eta_0 = 1.38$ .

#### 4. Results and discussion

Guide wall geometry has significant effect on the flow pattern; in other words, the flow formation at the entrance of approach channel is severely affected by the guide wall geometrics. Choosing the optimal geometry plays an important role in the spillway's performances; therefore, each seething and non-uniformity in the approach flow causes reduction of the spillway's capacity. The discharge coefficient of the spillways is important parameter reduced in the non-uniformity condition of approached flow and the risk of cavitation occurrence is increased as well. Optimal design of the guide wall geometry causes removing the non-uniformity flows and cross waves at the entrance of guide walls, through the approach channel and along the chute of spillway. To choose an optimal form for the Balaroud dam guide wall, three scenarios in which their geometric properties are shown in Figs. 3–5 were evaluated using the physical and numerical modeling. For optimal design of the guide wall in addition to show the flow pattern, the head

discharge curve is recorded to assess the performances of the each scenario as well. For this purpose as mentioned in the past the Flow 3D was used for modeling the flow pattern in the approach channel and over spillway as well. Fig. 8 shows a schematic figure of the CFD simulation of flow through the guide wall and over the spillway. To assess the performance of turbulence model three types of existing turbulence models including K-epsilon, LES and RNG-K-epsilon in Flow 3D were assessed. During the CFD simulation found that the RNG-K epsilon has better performance than the others for modeling of the flow through the guide wall.

Results of the numerical modeling of flow through the mode (1) are shown in Fig. 9 and its rating curve is shown in Fig. 10 (see Table 1).

Numerical modeling of the flow pattern through the guide wall model (1) shows that the geometric of the guide wall causes severely flow separation especially at the end of the direct guide wall and entrance of the approach channel, so this model is not suitable scenario. Numerical modeling results of flow pattern through the guide wall mode (2) are shown in Fig. 11. The head discharge data of the model (2) are presented in Table 2 similar to the model (1). The flow pattern (Fig. 11) through the guide wall shows that there is few deviance in the flow vectors but Fig. 12 shows that this shape for guide wall causes cross wave through the approach channel and spillway.

Assessing the performance of the mode (3) shows that this scenario has suitable performance for smoothly flow passing through the guide wall and this plan for the guide wall leads to remove the cross waves. Table 3 presents the rating curve data for the model (3). The results of numerical modeling of flow pattern and rating curve of the mode (3) are shown in Figs. 13 and 14.

As seen from Fig. 13, geometry of the model (3) causes the minimum separation along the guide wall and approach channel. In the following the flow cross section was plotted to evaluate the mode (3) performance to create non-uniformity and cross wave through the approach channel. As shown in Figs. 13 and 15, this type of the guide wall geometry causes smoothly flow passing with minimum cross wave creation. Fig. 16 compares the capacity of the three models for passing the flow. As shown in this figure models 1 and 2 have similar performance and the model (3) is more efficient. This model creates less flow depth for passing the flow discharge compared with other models. Increasing the performance of the model (3) is because of removing the cross waves and non-uniformity of flow through the guide wall and over the spillway. During the tur-

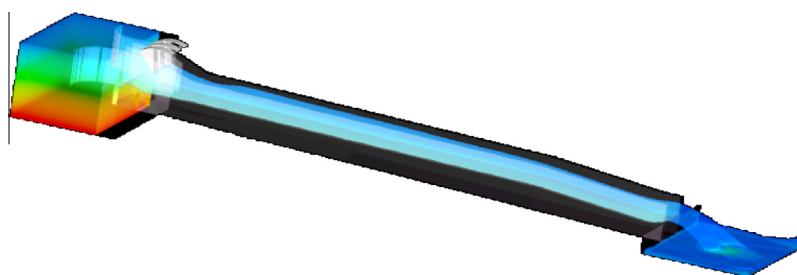


Figure 8 Schematic shape of the flow simulation through the guide wall and spillway.

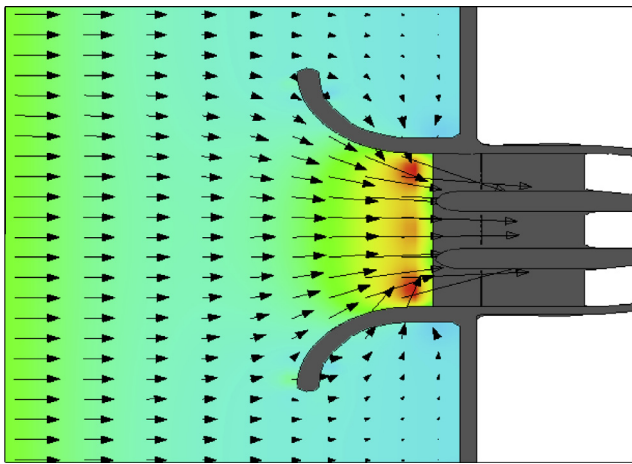


Figure 9 Flow pattern through the guide wall model (1) at return period 1000 year.

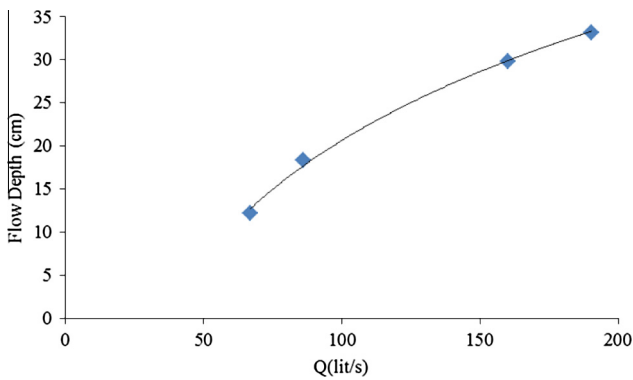


Figure 10 Rating curve of the model (1).

Table 1 The rating curve of the model (1).

| Return period (year) | $Q$ ( $m^3/s$ ) | Head (m) |
|----------------------|-----------------|----------|
| 2                    | 0.0667          | 0.122    |
| 100                  | 0.0859          | 0.1832   |
| 1000                 | 0.160           | 0.2987   |
| 10,000               | 0.1903          | 0.3321   |

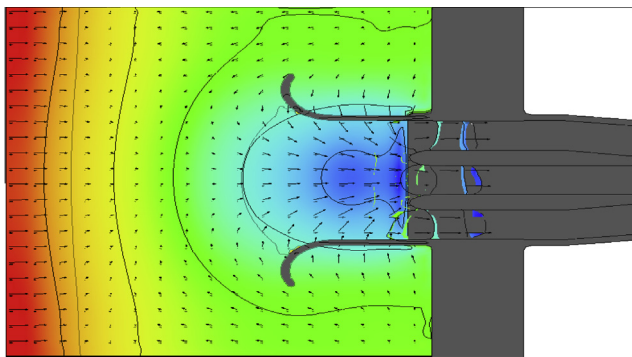


Figure 11 Flow pattern through the guide wall model (2) at return period 1000 year.

Table 2 Head discharge curve of the model (2).

| Return period (year) | $Q$ ( $m^3/s$ ) | Head (m) |
|----------------------|-----------------|----------|
| 2                    | 0.0667          | 0.122    |
| 100                  | 0.0859          | 0.1832   |
| 1000                 | 0.160           | 0.2987   |
| 10,000               | 0.1903          | 0.3321   |

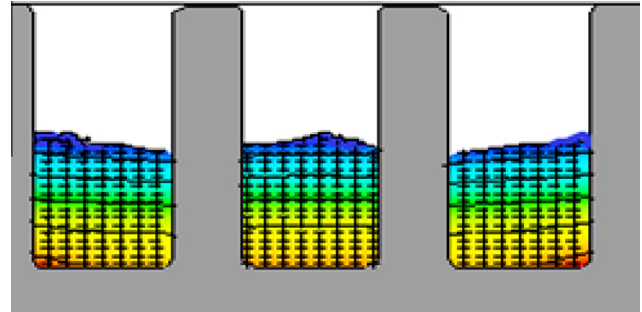


Figure 12 Cross section of the flow through the guide wall and cross wave on the spillway at flood return period 1000 year (model 2).

Table 3 Head discharge curve of the model (3).

| Return period (year) | $Q$ ( $m^3/s$ ) | Head (m) |
|----------------------|-----------------|----------|
| 2                    | 0.0667          | 0.1225   |
| 100                  | 0.0859          | 0.1614   |
| 1000                 | 0.160           | 0.2685   |
| 10,000               | 0.1903          | 0.3011   |

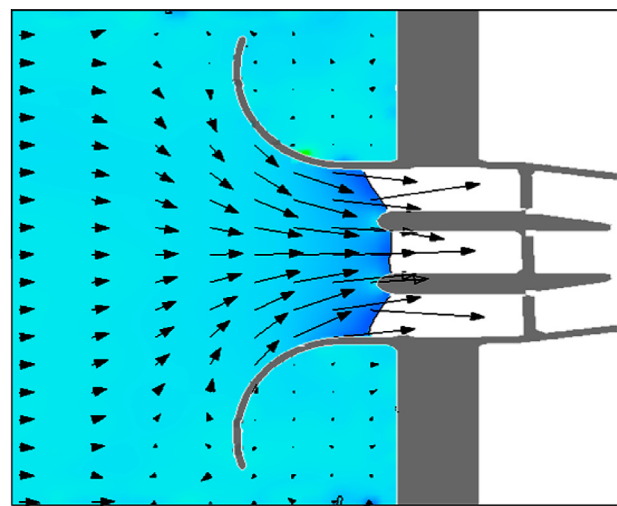


Figure 13 Flow pattern through the guide wall model (3) at return period 1000 year.

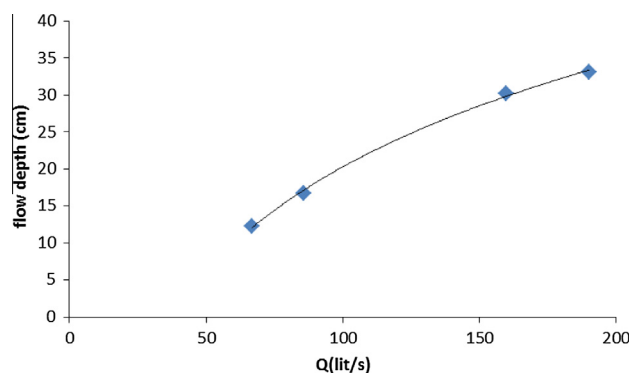


Figure 14 Head discharge curve of the model (3).

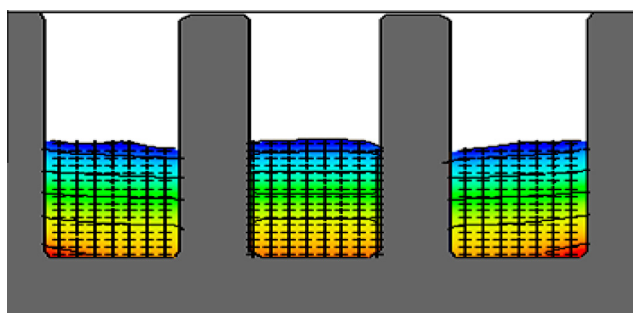


Figure 15 Cross section of the flow through the guide wall at the flood return period 1000 year.

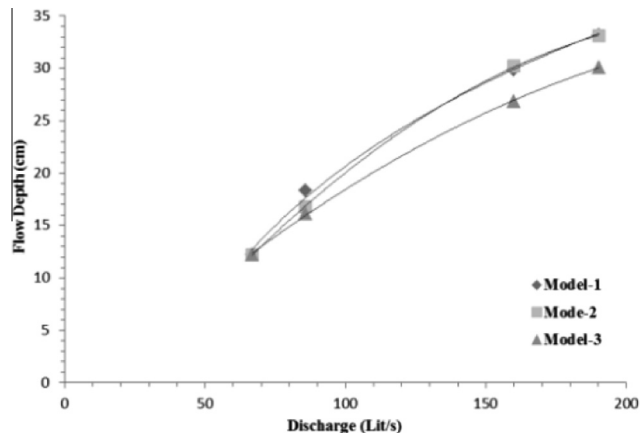


Figure 16 The rating curve of the models for the guide wall.

bulence models the RNG-K-epsilon has higher ability performance for modeling the flow pattern in comparison with the other models.

## 5. Conclusion

Numerical modeling of the hydraulic structure has become unavoidable works in the hydraulic engineering projects. Recently by advancing the CFD tools, the optimization of the structure was not limited to the structure size, whereas the shape of the structure geometries can be optimal which leads to increase the efficiency of the performance. In this

study three scenarios were considered for the guide wall dam. To assess these performances the CFD technique was used. Comparing these three models shows that the mode (3) is more efficient compared with others. Overall this study showed that optimal design of the guide wall leads to increase the performance of the spillway to pass the flow smoothly.

## References

- [1] M.C. Aydin, CFD simulation of free-surface flow over triangular labyrinth side weir, *Adv. Eng. Softw.* 45 (1) (2012) 159–166.
- [2] M.C. Aydin, M.E. Emiroglu, Determination of capacity of labyrinth side weir by CFD, *Flow Meas. Instrum.* 29 (2013) 1–8.
- [3] H. Babaali, A. Shamsai, H. Vosoughifar, Computational modeling of the hydraulic jump in the stilling basin with convergence walls using CFD codes, *Arab. J. Sci. Eng.* 40 (2) (2015) 381–395.
- [4] P.D. Bates, S.N. Lane, R.I. Ferguson, *Computational Fluid Dynamics: Applications in Environmental Hydraulics*, Wiley, 2005.
- [5] T. Cebeci, *Turbulence Models and Their Application: Efficient Numerical Methods with Computer Programs*, Horizons Pub, 2004.
- [6] P.G. Chanel, *An Evaluation of Computational Fluid Dynamics for Spillway Modeling*, University of Manitoba, 2008 (Master of Science).
- [7] J. Chatila, M. Tabbara, Computational modeling of flow over an ogee spillway, *Comput. Struct.* 82 (22) (2004) 1805–1812.
- [8] C. Chinnarasri, D. Kosittiwong, P.Y. Julien, Model of flow over spillways by computational fluid dynamics, *Proc. ICE – Water Manage.* (2014) 164–175.
- [9] R. Ettema, *Hydraulic Modeling Concepts and Practice*, ASCE, 2000.
- [10] D. Gessler, CFD modeling of spillway performance, *Impacts Glob. Clim. Change* (2005) 1–10.
- [11] G. Guyot, H. Maaloul, A. Archer, A Vortex modeling with 3D CFD, in: P. Gourbesville, J. Cunge, G. Caignaert (Eds.), *Advances in Hydroinformatics*, Springer, Singapore, 2014, pp. 433–444.
- [12] W.H. Hager, M. Pfister, *Hydraulic modelling – an introduction: principles, methods and applications*, *J. Hydraul. Res.* 48 (4) (2010) 557–558.
- [13] D. Kim, J. Park, Analysis of flow structure over ogee-spillway in consideration of scale and roughness effects by using CFD model, *KSCE J. Civ. Eng.* 9 (2) (2005) 161–169.
- [14] R. Maghsoodi, M.S. Roozgar, H. Sarkardeh, H.M. Azamathulla, 3D-simulation of flow over submerged weirs, *Int. J. Model. Simul.* 32 (4) (2012) 237.
- [15] B. Mohammadi, O. Pironneau, *Analysis of the K-epsilon Turbulence Model*, Wiley, 1994.
- [16] F. Moukalled, L. Mangani, M. Darwish, *The Finite Volume Method in Computational Fluid Dynamics: An Advanced Introduction with OpenFOAM® and Matlab*, Springer International Publishing, 2015.
- [17] Y. Nakayama, R.F. Boucher, *Computational fluid dynamics*, in: Y.N.F. Boucher (Ed.), *Introduction to Fluid Mechanics*, Butterworth-Heinemann, Oxford, 1998, pp. 249–273.
- [18] A. Parsaie, A. Haghiabi, Computational modeling of pollution transmission in rivers, *Appl. Water Sci.* (2015) 1–10.
- [19] A. Parsaie, A. Haghiabi, Predicting the longitudinal dispersion coefficient by radial basis function neural network, *Model. Earth Syst. Environ.* 1 (4) (2015) 1–8.
- [20] A. Parsaie, A. Haghiabi, A. Moradinejad, CFD modeling of flow pattern in spillway's approach channel, *Sustain. Water Resour. Manag.* 1 (3) (2015) 245–251.

- [21] A. Parsaie, H. Yonesi, S. Najafian, Predictive modeling of discharge in compound open channel by support vector machine technique, *Model. Earth Syst. Environ.* 1 (2) (2015) 1–6.
- [22] S. Patankar, *Numerical Heat Transfer and Fluid Flow*, Taylor & Francis, 1980.
- [23] C. Pozrikidis, *Fluid Dynamics: Theory, Computation, and Numerical Simulation*, Springer, US, 2009.
- [24] W. Rodi, *Turbulence Models and Their Application in Hydraulics*, Taylor & Francis, 1993.
- [25] M. Suprpto, Increase spillway capacity using labyrinth weir, *Proc. Eng.* 54 (2013) 440–446.
- [26] H.K. Versteeg, W. Malalasekera, *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*, Pearson Education Limited, 2007.
- [27] J.-B. Wang, H.-C. Chen, Improved design of guide wall of bank spillway at Yutang Hydropower Station, *Water Sci. Eng.* 3 (1) (2010) 67–74.
- [28] J. Wang, H. Chen, Experimental study of elimination of vortices along guide wall of bank spillway, in: *Advances in Water Resources and Hydraulic Engineering*, Springer, Berlin Heidelberg, 2009, pp. 2059–2063.