Scientia Iranica A (2015) 22(5), 1734–1741

SCIENTIA IRANICA

Sharif University of Technology
Scientia Iranica
Transactions A: Civil Engineering
www.scientiairanica.com

Research Note

Numerical simulation of turbulent flow over labyrinth spillways/weirs and corresponding discharge coefficient and efficiency

V. Zahraeifard* **and N. Talebbeydokhti**

Department of Civil & Environmental Engineering, Shiraz University, Namazi Sq., Zand Ave., Shiraz, Iran.

Received 29 May 2013; received in revised form 2 November 2014; accepted 26 May 2015

KEYWORDS
Labyrinth spillway;
Discharge coefficient;
Computational fluid dynamics;
Turbulence;
Free-surface;
Fluent.

Abstract. Labyrinth spillway has become an appropriate choice to increase an existing spillway's capacity. Thus, it is necessary to understand the hydraulic performance of labyrinth spillways/weirs. This paper numerically solves turbulent flow over labyrinth spillways/weirs and determines the discharge coefficients. Reynolds governing equations, turbulence $k-\varepsilon$ model, and the Volume Of Fluid (VOF) model are numerically solved to define pressure, velocity, and the free surface flow profiles. The numerical results are comparable to those obtained from physical modeling with maximum 6.43% error relative to results of physical modeling. Present study indicates that numerical simulation can be used to supplement physical modeling. Thus, by using numerical solutions, the site specifics of the spillway which are often different from the conditions of design curves can be investigated. Also, the effect of different shapes of apex on discharge capacity of the labyrinth spillway is investigated in this study. The analyses show that labyrinth spillways with round apex shape are the most efficient spillways. Numerical determination of free water surface is presented and discussed, which is helpful for optimum design of stilling basin and leading walls.

1. Introduction

A vital component of the dam's design is the spillway. Spillways are hydraulically designed to pass excessive water that cannot be contained in the allotted storage space. Many dam failures have been caused by improperly designated spillways or by spillways of insufficient capacity [1]. In this regard, discharge coefficient of the spillway is the most important parameter, which is dependent on water head over crest, crest shape, and crest length. Improvements in the fields of meteorology and hydrology have gained attention among researchers and engineers to re-evaluate the performance of spillways as it is directly affected by updated hydrological data [2]. In fact, a reliable flood passage capacity of the spillway is crucial. If an existing spillway has inadequate capacity to pass the maximum design flood, there are three alternatives for remedy: 1) Increase the volume of dam storage; 2) Increase the spillway's capacity; and 3) A combination of these two options [2]. The main concern with the second alternative is accessible space to widen the spillway's length. This concern can be resolved technically and economically by using labyrinth spillways. Several physical studies have been conducted on labyrinth spillways/weirs. Hay and Taylor (1970) evaluated various performance modes of labyrinth weirs, the effect of

*. *Corresponding author. Tel.: +98 71 36287505;*
Fax: +98 71 36286619
E-mail addresses: v.zahraee@gmail.com (V. Zahraeifard); taleb@shirazu.ac.ir (N. Talebbeydokhti)

bottom canal slope on overflow capacity of labyrinth weirs, and the influence of water head over the weir's crest [3]. Their study provides performance curves (design curves) under different conditions. Darvas (1971) provided a definition for the discharge coefficient of labyrinth spillways [4]. Houston and Hinchliff (1982) summarized the results of physical modeling of labyrinth spillways for Hyrum dam by the U.S. Bureau of Reclamation (USBR) [5]. In 1985, Cassidy et al. examined the performance of labyrinth spillways for high water head and found a 20% reduction in efficiency of the labyrinth spillways [6]. Lux and Hinchliff (1985) accomplished dimensional analysis to get discharge coefficient of labyrinth spillways and presented design curves [2]. Magalhães and Lorena (1989) introduced a formulation for determination of discharge coefficient along with design curves [7]. Finally, Tullis et al. (1995) conducted a comprehensive study to evaluate the discharge coefficient, effect of water head, and angle of side walls (Figure 1) [8]. They introduced their design curves. Lopes et al. (2008) studied flow conditions downstream of labyrinth spillway in laboratory [9]. Khode and Tembhurkar (2010) conducted extensive physical modeling of labyrinth spillways [10] to evaluate design method introduced by Lux (1984) [11] and Tullis et al. (1995) [8]. Crookston (2010) conducted laboratory experiments to get discharge coefficients for quarter-round and half-round labyrinth weir with side angle between 6° to 35° [12]. Although several performance curves have been proposed for labyrinth spillways that can be used in new projects, these curves usually consider ideal conditions; e.g. assuming the approaching flow is perpendicular to the spillway [13]. In addition, the site specifics of spillways vary from idealized condition so that design curves are no longer applicable [13]. In other words, disturbing the ideal conditions of design curves disqualifies the validation of them. Furthermore, conducting physical studies for new projects by considering site-specifics is not economically feasible [14] and is time consuming as well. In recent years, advancements in computing power and Computational Fluid Dynamics (CFD) algorithms have resulted in evolving new tools for evaluation of different flow conditions and different design alternatives [15]. Savage et al. (2004) and Danish Hydraulic Institute (DHI) (2005) presented numerical simulation of flows past labyrinth spillways [15,16]. However, these studies were confined to specific projects without considering the effect of apex shape.

The objective of this study is to numerically study flows over labyrinth spillways/weirs using the Reynolds Averaged Navier-Stokes equations. Due to turbulent flow, the RNG $k-\varepsilon$ model is used. In addition, a two-phase flow (water and air) is considered over the labyrinth spillway which is important to define variation of free-surface. Thus, the robust Volume Of Fluid (VOF) method was used to determine the location and orientation of the interface between the water and air (free-surface). The specific objective of present study is to evaluate the effect of the apex shape on discharge capacity of labyrinth spillways.

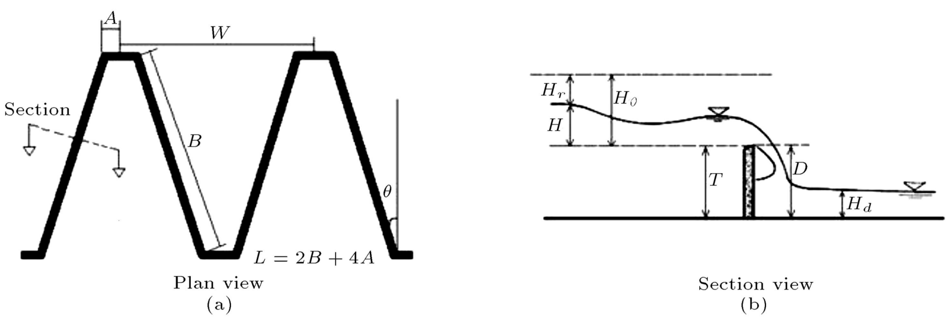
2. Materials and methods

2.1. Labyrinth spillways/weirs

As stated by Hay and Taylor (1970), a labyrinth weir is characterized by broken axis in plan so that the water flow is over a greater length of crest compared to a normal weir occupying the same lateral space [3]. Figure 1 shows a sketch of labyrinth spillway/weir, related parameters, and section view of flow over the labyrinth weir/spillway.

2.2. Governing equations

The governing equations of flow over wires are the well-known Navier-Stokes equations; one continuity equation and three momentum equations for the three coordinate directions. By including the effect of tur-



θ: Angle between wall and flow direction
W: Width of weir in a cycle
L: Length of crest in a cycle
A: Half of the apex length
B: Length of lateral wall
T: Weir height at upstream
D: Weir height at downstream (in this figure is equal to T)
H: Piesometric head at upstream
H_r: Velocity head at upstream
H_d: Piesometric head at downstream
H_o: Total head over the crest

Figure 1. Labyrinth weir and the corresponding parameters: (a) Plan view; and (b) section view of flow over labyrinth weir.

bulence, the equations change to Reynolds equations as [17,18]:

Continuity equation:

$$\frac{\partial U_i}{\partial x_i} = 0, \tag{1}$$

Momentum equation:

$$\rho\frac{\partial U_i}{\partial t}+\rho\frac{\partial}{\partial x_j}\left(U_jU_i+\overline{u_j'u_i'}\right)=-\frac{\partial P}{\partial x_i}+\frac{\partial}{\partial x_j}(2\mu S_{ji}), \tag{2}$$

in which U_i is average velocity in (i) direction; P is pressure; μ is molecular viscosity; ρ is density; s_{ji} is strain-rate tensor; and $\overline{u_j'u_i'}$ is Time-averaged momentum due to turbulence which is called Reynolds stresses. $\rho\overline{u_i'u_j'}$ has nine components [17]:

$$\rho\overline{u_i'u_j'} = \begin{bmatrix} \rho(\overline{u_1'^2}) & \rho\overline{u_1'u_2'} & \rho\overline{u_1'u_3'} \\ \rho\overline{u_2'u_1'} & \rho(\overline{u_2'^2}) & \rho\overline{u_2'u_3'} \\ \rho\overline{u_3'u_1'} & \rho\overline{u_3'u_2'} & \rho(\overline{u_3'^2}) \end{bmatrix}. \tag{3}$$

These unknowns due to turbulence are determined by using turbulence models. These models consist of semi-empirical equations which relate the fluctuating components of quantities to the average components. The most popular turbulence model is $k-\varepsilon$ model. The RNG scheme of this model was used to solve Reynolds stresses.

2.3. Volume Of Fluid (VOF) approach

This method was introduced first by Hirt and Nichols in 1981 [19]. The fundamental concept of this method for multi-phase flow is to determine the fraction of computational cells occupied by each fluid and to determine where the interface between fluids occurs. The location of the interface is calculated on the basis of the following equations [18,20]:

$$\frac{\partial\alpha_w}{\partial t} + u_i\frac{\partial\alpha_w}{\partial x_i} = 0, \tag{4}$$

$$\alpha_a = 1 - \alpha_w, \tag{5}$$

in which α_w and α_a are, respectively, the fractions of water and air within a cell. The above equations are for a two-phase flow which consists of water and air; the subscript "w" refers to water and "a" refers to air. Eq. (5) indicates that the summation of all fractions equals unity. In fact, calculating α_w and α_a in all cells provides information about the location of interface. The momentum equation for this two-phase flow is like that of a single flow expressed by Navier- Stokes equations. However, ρ (density) and μ (molecular viscosity) should be modified due to the variations of each fluid fraction. Thus, we have:

$$\rho = \alpha_w\rho_w + (1-\alpha_w)\rho_a. \tag{6}$$

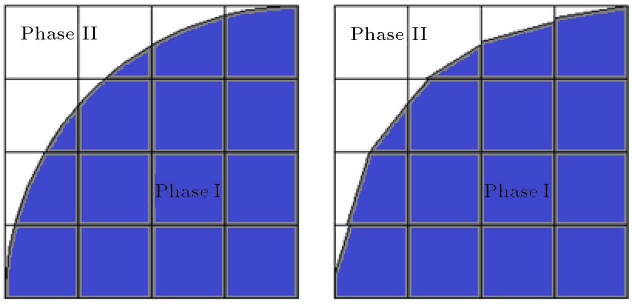


Figure 2. Interface formation between two phases of fluid: (a) Actual interface; and (b) interface predicted by using geometric reconstruction scheme. Re-produced based on work of Tang et al. [21].

$$\mu = \alpha_w\mu_w + (1-\alpha_w)\mu_a, \tag{7}$$

in which ρ_a and ρ_w are density of air and water, while μ_a and μ_w are the molecular viscosity of air and water, respectively. To determine the orientation of the interface between the fluids, several algorithms have been introduced. The most accurate one is called the Geometric Reconstruction Scheme which consists of two steps: 1- Reconstruction 2- Convection. The main goal of this reconstruction algorithm is to clarify the orientation of segments in each computational cell. This is done by determination of normal unit vector of each segment within a cell. α_w in neighboring cells of cell (ij) are used to get normal unit vectors. Then, the angle between positive direction of x-axis and the interface of fluids can be defined (β). Figure 2 shows schematically how accurate the Geometric Reconstruction Scheme predicts the orientation of the interface in the bulk of a two-phase flow [21]. Therefore, free water surface can be predicted for flow over labyrinth spillway.

$$\begin{cases} n_{ij}^x = \frac{1}{\Delta x}\Big(\alpha_{wi+1j+1} + 2\alpha_{wi+1j} + \alpha_{wi+1j-1} \\ \qquad -\alpha_{wi-1j+1} - 2\alpha_{wi-1j} - \alpha_{wi-1j-1}\Big) \\ n_{ij}^y = \frac{1}{\Delta y}\Big(\alpha_{wi+1j+1} + 2\alpha_{wij+1} + \alpha_{wi-1j+1} \\ \qquad -\alpha_{wi+1j-1} - 2\alpha_{wij-1} - \alpha_{wi-1j-1}\Big) \end{cases} \tag{8}$$

$$\beta = \tan^{-1}\left(\frac{-n^x}{n^y}\right). \tag{9}$$

Thus, by VOF, the movement of the interface is appropriately modeled.

2.4. Numerical solution

Fluent is a general solver of flow field in 2D and 3D [18]. The first editions of fluent were able to only analyze problems with structured mesh. Now, it can analyze both structured and unstructured meshes.

Analyzing the incompressible flow or heat transfer with fluent is very common. But, this simulation platform has seldom been used for free surface problems like those that arise in civil engineering. This code solves the presented governing equation to define the flow field. Including both turbulence models and VOF model is of great importance to utilize this code. To solve the partial differential governing equations, fluent employs the finite volume method. Discretization of the governing equations can be done by applying the upwind method. The simulated velocity and pressure fields are coupled by using 'SIMPLE method' [18]. To get the numerical solution to Eqs. (1) to (9), the models are prepared in Gambit which is a mesh generator. Unstructured meshes were used to produce the computational cells. In mesh generation, care should be taken where the rate of gradients is high and therefore a larger number of cells are needed. A preliminary study defined the appropriate mesh sizes. The size of mesh along all coordinates was selected 2 cm except for high gradient zones where the mesh size was reduced to 8-9 mm. Suitable boundary conditions are of primary importance. These conditions are shown in Figure 3. A "water inlet: pressure inlet" boundary condition was used to define the water pressure at flow inlets. This boundary condition is based on the assumption that upstream inlet is sufficiently far away from the crest where velocity is negligible. As a result, "water inlet: pressure inlet[[defines hydrostatic pressure of water column at different heights. Similarly, there is also "air inlet: pressure inlet" boundary condition at the top of water surface which defines atmospheric pressure. For the leading walls at two sides of flow as well as for the bottom, the "wall" boundary condition is assigned to bound fluid and solid regions. For downstream, a "pressure outlet" boundary was considered to specify static pressure at outlet. For present study, due to steep slope section downstream of weir, water depth was negligible. Thus, static pressure at downstream is close to atmospheric pressure. Initial conditions are also necessary to start the numerical solution of the equations. The initial conditions are hydrostatic pressure on the weir/spillway walls, bottom, and water inlet at upstream of labyrinth weir/spillway. A column of water is assumed right above the crest before the numerical simulation begins.

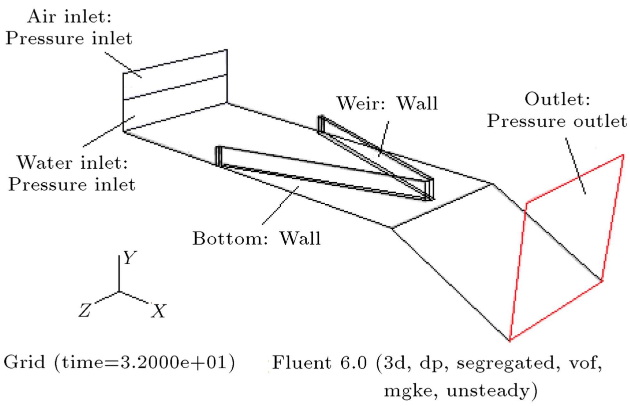


Figure 3. Boundary conditions used in simulation throughout the domain.

As numerical simulation continues, discharge of flow is calculated as the product of flow velocity and cross section. Discharge coefficient can be obtained according to Eq. (10).

$$Q = \frac{2}{3} C_d L \sqrt{2g} H_o^{1.5}. \tag{10}$$

In Eq. (10), L and H_o are the effective length of weir and total head on the crest, respectively. L is defined according to the geometry of the weir/spillway, and total head (H_o) is defined as the summation of static head ($P/\rho g$) and velocity head ($U^2/2g$) that are determined once the pressure and velocity fields are calculated by numerical solution. When discharge is obtained by numerical simulation, C_d is the only unknown of Eq. (10).

2.5. Geometric characteristics of the present testing weirs

The numerical study includes two sets of models: Set (I) and set (II). The models in set (I) are numerical simulations of the labyrinth spillway previously used in the laboratory experiments by Tullis et al. (1995) in the hydraulic laboratory at the Utah Water Research Laboratory at Utah State University [8]. The only difference, which does not have a significant effect on the results, is the width of apex. Tullis et al. (1995) removed the width [8] and therefore considered triangular configurations for their tested models while, in practice, a minimum width should be considered. The sketches of the weirs for set (I) with the important parameters are shown in Figure 4 and Table 1. For the models of set (I), the effect of various water heads are evaluated. In the set (II) models, the effect of apex

Table 1. The parameters of weirs set (I).

θ (degree)	Length of weir (m)	Width of weir (m)	T (m)	L/W	W/T
12	4.512	1	0.152	4.512	6.579
18	2.909	1	0.152	2.909	6.579
35	1.666	1	0.152	1.666	6.579

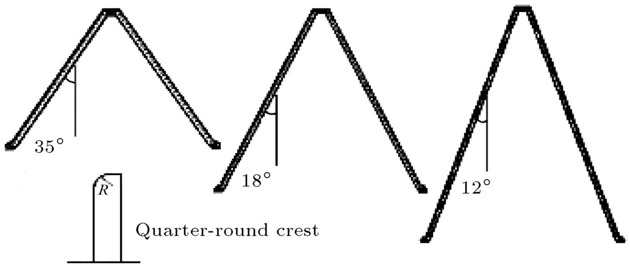


Figure 4. The sketch of weirs of set (I).

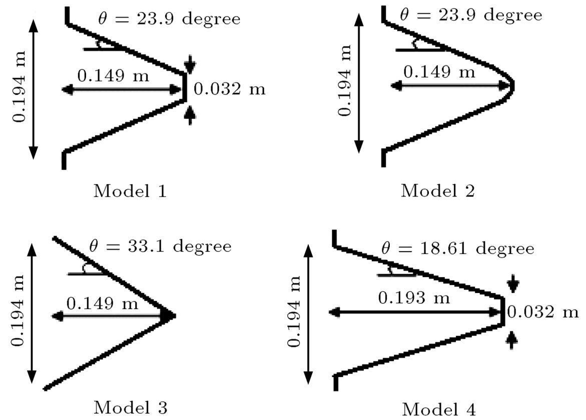


Figure 5. The sketch of weirs in set (II).

shape is taken into consideration (Figure 5). In this set of models, labyrinth weirs with a wide apex are defined by model nos. 1 and 4, with a round apex by model no. 2, and with a sharp apex by no. 3. For all of the models, a steep-slope section after the spillway helps easy discharge of water and prevents submergence to affect the discharge coefficient of flow over the weir/spillway.

3. Results

The results of numerical study along with the physical results for models in set (I) are presented in Table 2. Numerical computation was continued until steady state flow occurred. The errors between results of physical and numerical models are given in Table 3 to indicate performance of numerical simulation. Figures 6 and 7 depict the pressure and velocity fields over the labyrinth weir with $\theta = 18°$ and $H_o/T = 0.8$. The pressure field in Figure 6 is static pressure rather than total pressure. This is mainly due to negligible

Table 3. Percentage error* in results of numerical study.

$H_o/T = 0.25$	$H_o/T = 0.5$	$H_o/T = 0.8$
–	–	-4.237
6.43	<1	<1
–	<1	-4.42

* Numerical error (%)$=\frac{\left(\frac{H_o}{T}\right)_{\text{numerical}}-\left(\frac{H_o}{T}\right)_{\text{physical}}}{\left(\frac{H_o}{T}\right)_{\text{physical}}}\times 100.$

Table 4. Efficiency of labyrinth weirs with different apex shape.

Model no.	$Q_L \times 10^{-3}$ (m^3/s)	Q_L/Q_N
1	29.4	47.1
2	62.4	586.1
3	21.4	443.1
4	36.5	8.1

flow velocity (in the order of 10^{-3} to 10^{-1} m/s) in upstream domain and even on weir's crest which produces negligible dynamic pressure ($\rho U^2/2$). The free surface of flow for the same labyrinth weir is shown in Figure 8. After getting suitable results for the models of set (I), the effect of apex shapes was examined based on models in set (II). The efficiencies of models in set (II) are given in Table 4. Efficiency is defined as the ratio of flow discharge over labyrinth weir (Q_L) to the flow discharge (Q_N) over a straight weir with the same width or (Q_L/Q_N).

4. Discussion

Simulation of flow over labyrinth spillways/weirs was presented in this paper. Due to 3D nature of flow over labyrinth spillways, almost all of the previous investigations have been accomplished experimentally. In fact, in those investigations, the lack of powerful computers and sophisticated algorithms which are capable of both defining free-water-surface and computation of hydrodynamic parameters (velocity and pressure) prevent application of numerical modeling for study of overflow on labyrinth spillways. As shown in this paper, through numerical modeling (Tables 2 and 3), the errors between physical and numerical modeling are generally below 10%. In cases in which labyrinth weir is with $\theta = 18°$ and $H_o/T = 0.25$ in set (I), the error is relatively higher; the impact of reaeration

Table 2. Results of numerical simulation for cases of set (I).

θ	C_d based on the study of Tullis et al. 1995			C_d based on numerical study		
(degree)	$H_o/T = 0.25$	$H_o/T = 0.5$	$H_o/T = 0.8$	$H_o/T = 0.25$	$H_o/T = 0.5$	$H_o/T = 0.8$
12	0.551	0.437	0.348	–	–	0.333
18	0.622	0.554	0.459	0.662	0.551	0.457
35	0.712	0.706	0.653	–	0.706	0.624

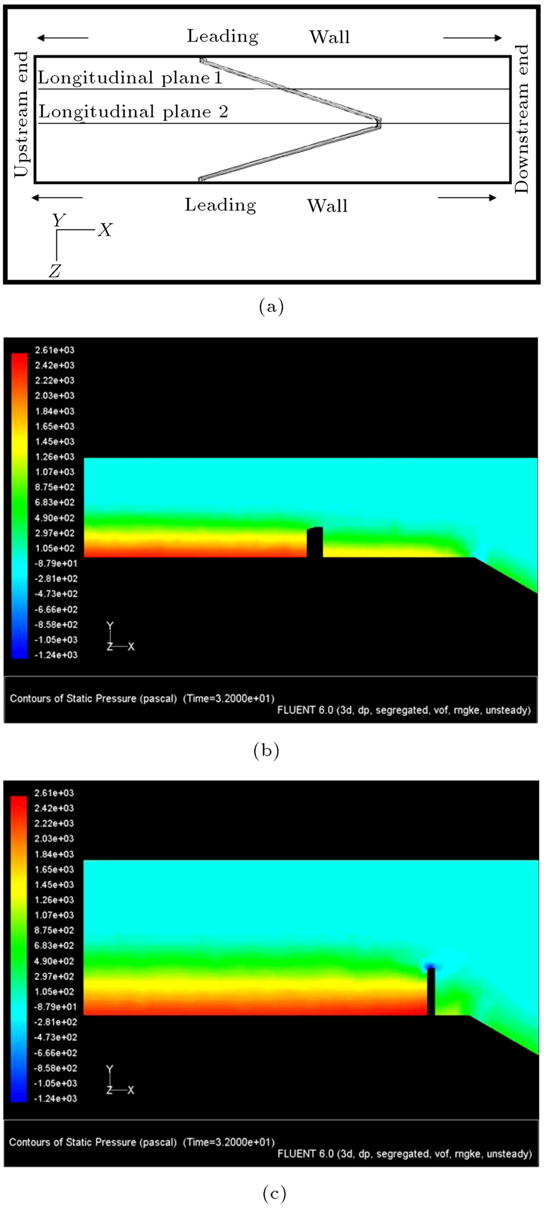


Figure 6. Pressure field in terms of static pressure for labyrinth weir with $H_o/T = 0.8$: (a) Location of longitudinal planes for presenting pressure field; (b) pressure field along longitudinal plane 1; and (c) pressure field along longitudinal plane 2.

condition is probably the source of difference as also pointed by Tullis et al. (1995) [8]. It is clear from Table 4 that the efficiency of labyrinth weir no. 1 in set (II) is greater than labyrinth weir no. 3 since L/W ratio is higher for labyrinth of no. 1. In addition, by sharpening the apex of labyrinth spillways/weirs, the efficiency of labyrinth increases. Thus, the efficiency of labyrinth no. 2 is greater than labyrinth no. 1. The results in Table 4 also show that smaller angle (θ) causes more efficiency of the labyrinth weir. Therefore, the efficiency of weir no. 4 is greater than efficiency of labyrinth weir no. 1.

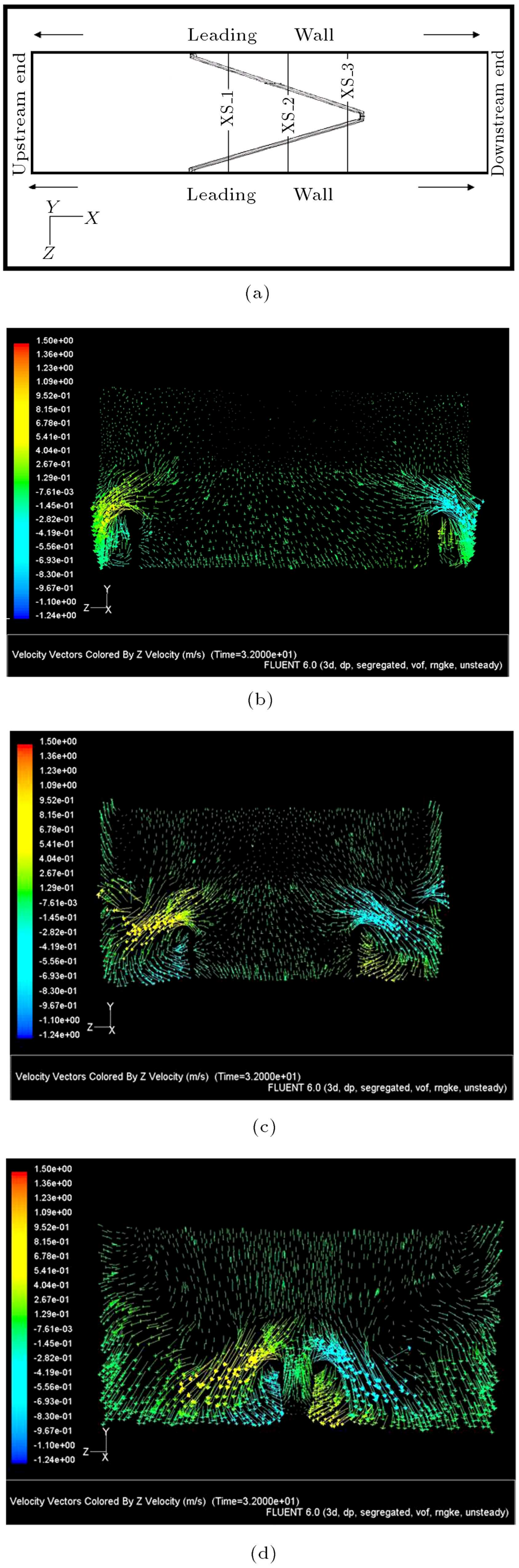


Figure 7. Velocity vectors over labyrinth weir with $H_o/T = 0.8$: (a) Location of cross-sections (XS_1, XS_2, XS_3) for presenting velocity field; (b) velocity vectors at XS_1; (c) velocity vectors at XS_2; and (d) velocity vector at XS_3.

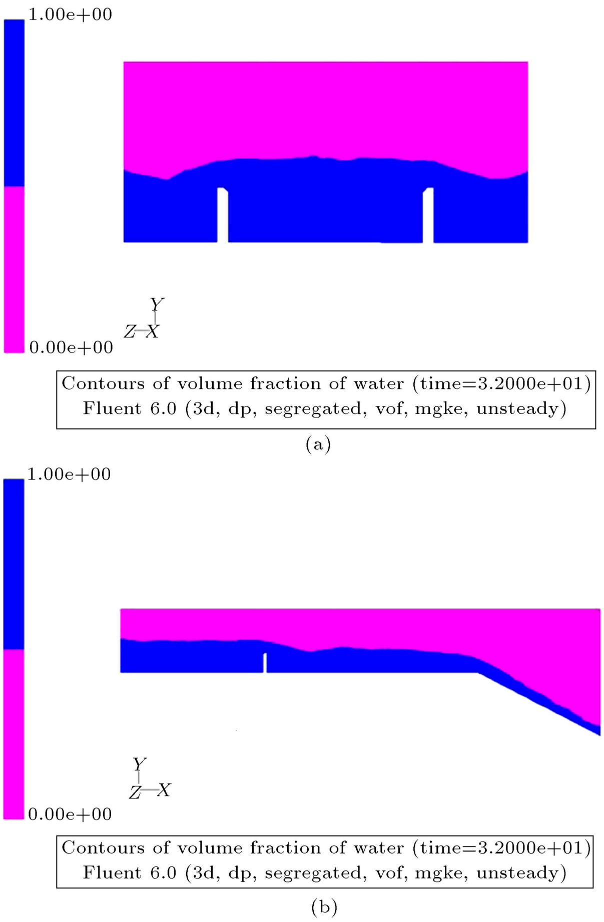


Figure 8. Free water surface over labyrinth weir with $H_o/T = 0.8$: (a) Perpendicular to flow path; and (b) parallel to flow path.

Numerical study of flow over labyrinth spillways yields the value of velocity and pressure for the entire computational domain. Thus, by defining appropriate domain, the velocity of flow in the stilling basin right downstream of spillways can also be obtained. This velocity is important for determination of dimensions of stilling basin. In addition, vibration of the nappe that produces loud noise is very much dependent on variation of pressure which can be fully analyzed prior to the construction of the spillway. As shown in Figure 8, VOF model is able to identify free-surface throughout the computational domain. It is clear from Figure 8 that water surface slightly falls where it reaches contraction and increase while exiting the downstream apex. This variation in water surface is also depicted in Figure 1(b) in section view of flow over labyrinth weir. Water surface elevation can be used in economic determination of leading walls' height.

5. Conclusion

Re-evaluation of the workability of different parts of dams especially spillways on the basis of updated hydrologic and meteorological data is of primary importance. In this regard, application of labyrinth spillways due to increasing volume of dam storage and discharge capacity has been highly considered. The results of this study show:

- Numerical simulation is a suitable tool for evaluation of flow over labyrinth spillways under different geometries of spillway and various hydraulic conditions.
- Instead of using design curves from literature, numerical modeling can be directly used in design of new spillways by incorporating site-specifics which are usually different from conditions of design curves.
- The results from numerical studies can be used to obtain optimum design of spillway, stilling basin, and leading walls.

Acknowledgment

The authors would like to extend their sincere acknowledgment to the reviewers for their insightful comments and corrections and improving present submission.

References

1. Mays, L.W. "Water resources engineering", Chap. 17, Wiley, New York, USA (2001).
2. Lux, F. III and Hinchliff, D.L. "Design and construction of labyrinth spillways", *15th ICOLD Conference*, Lausanne, Switzerland, pp. 249-274 (1985).
3. Hay, N. and Taylor, G. "Performance and design of labyrinth weirs", *ASCE J. of Hydr. Eng.*, **96**(11), pp. 2337-2357 (1970).
4. Darvas, L.A. "Discussion of performance and design of labyrinth weirs by Hay and Taylor", *ASCE J. of Hydr. Eng.*, **97**(80), pp. 1246-1251 (1971).
5. Houston, K.L. and Hinchliff, D.L. "Hydraulic model study of hyrum dam auxiliary labyrinth spillway", Report No GR-82-13, U.S. Dept. of the Interior, Bureau of Reclamation (1983).
6. Cassidy, J.J., Gardner, Ch. A. and Peacock, R.T. "Boardman labyrinth crest spillway", *ASCE J. of Hydr. Eng.*, **111**(3), pp. 398-416 (1985).
7. Magalhães, A. and Lorena, M. "Hydraulic design of labyrinth weirs", Report No. 736, National Laboratory of Civil Engineering, Lisbon, Portugal (1989).
8. Tullis, J.P., Amanian, N. and Waldron, D. "Design of labyrinth spillways", *ASCE J. of Hydr. Eng.*, **121**(3), pp. 247-255 (1995).
9. Lopes, R., Matos, J. and Melo, J.F. "Characteristic depths and energy dissipation downstream of a labyrinth weir", *Proc. of the Int. Junior Researcher*

and Engineer Workshop on Hydraulic Structures, S. Pagliara, Ed., Edizioni Plus, University of Pisa, Pisa, Italy, pp. 51-58 (2008).

10. Khode, B.V. and Tembhurkar, A.R. "Evaluation and analysis of crest coefficient for labyrinth weir", *World Appl. Sci. J.*, **11**(7), pp. 835-839 (2010).

11. Lux III, F. "Discharge characteristics of labyrinth weirs", *Proc. ASCE Hydr. Div. Specialty Conf.*, New York, USA (1984).

12. Crookston, B.M. "Labyrinth weirs", Ph.D. Dissertation, Utah State University, Logan, Utah. (2010).

13. Falvey, H.T. *Hydraulic Design of Labyrinth Weirs*, ASCE Press, Reston, VA, USA (2003).

14. Ho, D.K.H., Boyes, K.M. and Donohoo, S.M. "Investigation of spillway behavior under increased maximum flood by computational fluid dynamics technique", *14th Australasian Fluid Mechanics Conference*, Adelaide University, Adelaide, Australia (2001).

15. Savage, B.M., Frizell, K. and Crowder, J. "Brains versus Brawn: The changing world of hydraulic model studies", *Proc. of the ASDSO Annual Conference*, Phoenix, AZ, USA (2004).

16. Danish Hydraulic Institute website (DHI) <http://ballastwater.dhigroup.com/-/media /publications/news/2009/070509ns3.pdf>. (2010).

17. Wilcox, D.C., *Turbulence Modeling for CFD*, DCW Industries, Inc., La Canada, CA, USA (2006).

18. Fluent 6.0 Documentation, "FLUENT user's guide", Fluent Inc., Lebanon, NH, USA (2001).

19. Hirt, C.W. and Nichols, B.D. "Volume of fluid (VOF) method for the dynamics of free boundaries", *J. of Comput. Phys.*, **39**(1), pp. 201-225 (1981).

20. Chen, Q., Dai, G. and Liu, H. "Volume of fluid model for turbulence numerical simulation of stepped spillway overflow", *ASCE J. of Hydr. Eng.*, **128**(7), pp. 683-688 (2002).

21. Tang, H., Wrobel, L.C. and Fan, Z. "Tracking of immiscible interfaces in multiple-material mixing process", *Comp. Mater. Sci.*, **29**(1), pp. 103-118 (2004).

Biographies

Vahid Zahraeifard is a PhD student in the Department of Civil and Environmental Engineering at Louisiana State University, Baton Rouge, USA. His fields of interest are: ecohydrology, numerical models in water resources engineering, surface water-groundwater interactions, fate and transport of solutes in aquatic environments. He holds two Master's degrees in Civil and Environmental Engineering from Louisiana State University and Shiraz University in 2011 and 2006, respectively. He received his Bachelor's degree in Civil Engineering from University of Tabriz in 2003.

Nasser Talebbeydokhti is a Professor of Civil and Environmental Engineering at Shiraz University with 30 years of teaching, research, and consulting activities that cover broad areas of water resources and environmental engineering. These include: environmental engineering, hydrology, river hydraulics, watershed engineering, sediment transport and channel morphology, hydraulic structures, hydropower, fish habitat and fish passage engineering, environmental impact assessment and mitigation, water resources planning, water quality management and monitoring, integrated watershed management, coastal and estuarine sediment, and river resource planning and management. Prof. Talebbeydokhti received his PhD and Master's degrees in Civil and Environmental Engineering from Oregon State University, Corvallis, USA, in 1979 and 1984, respectively. He also holds a Bachelor's degree in Water and Irrigation Engineering from Tehran University in 1974.