Scientia Iranica A (2014) 21(1), 91–97

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

Research Note

Numerical simulation of flow over spillway based on the CFD method

E. Fadaei-Kermani* **and G.A. Barani**

Department of Civil Engineering, Shahid Bahonar University, Kerman, P. O. Box 76169133, Iran.

Received 31 August 2012; received in revised form 25 April 2013; accepted 13 August 2013

KEYWORDS
Spillway;
Numerical modeling;
CFD method;
RNG model;
Flow characteristics.

Abstract. In this study, numerical simulation of flow over a chute spillway is presented using the Computational Fluid Dynamics (CFD) method. The flow characteristics such as velocity, pressure and depth through the spillway have been calculated for four different flow rates. Since the actual flow is turbulent, the RNG turbulence model has been used for simulation. The numerical computed results of piezometric pressure and flow velocity along the spillway were compared with the results from the hydraulic model tests. The maximum difference between calculated and experimental results in average velocity values was 5.47% and in piezometric pressure values was 7.97%. The numerical results agreed well with experiments.

1. Introduction

A spillway is one of the most important hydraulic structures designed to prevent overtopping of dams and provide sufficient safety and stability during floods. Improper design of spillways may cause failure in dams; consequently, the spillway must be carefully designed and hydraulically adequate to verify the flow characteristics [1].

Analysis of flow over a spillway can be an important engineering problem. Therefore, recent developments in computer science and numerical techniques have advanced the use of Computational Fluid Dynamics (CFD) as a powerful tool for this purpose.

Computational Fluid Dynamics (CFD) is a type of numerical method used to solve problems involving fluid flow. Since CFD can provide a faster and more economical solution than physical models, engineers are interested in verifying the capability of CFD software [2]. Some recent works show the capability of the CFD method in the numerical modeling of flow over several spillways. Savage and Johnson [3] simulated flow over a standard ogee-crest spillway using a commercial CFD code (flow-3D). They found good agreement between the physical and numerical models for both pressures and discharges. Ho et al. [4] described the two- and three-dimensional CFD modeling of spillway behavior under rising flood levels. The results were validated using published data, and acceptable results were obtained. Kim and Park [5] investigated flow characteristics on the ogee spillway using flow-3D software. They found that the scale effects on the model are in an acceptable error range if the length scale ratio is less than 100 or 200. Dargahi [6] did a three- dimensional simulation of flow over a spillway by means of CFD software (Fluent). He predicted the water surface profiles and the discharge coefficients for a laboratory spillway within an accuracy range of 1.5-2.9%, depending on the spillway's operating head. Zhenwei et al. [7] simulated flow over the whole spillway based on the VOF model using Fluent software. They found reasonable agreement between numerical and experimental results for the surface elevation, pressure and flow velocity along the spillway.

*. *Corresponding author.*
E-mail address: ehsanhard@gmail.com (E. Fadaei)

Table 1. Some features of the Shahid Abbaspour dam.

Type	Height (m)	Length (m)	Reservoir capacity (m^3)	Type of spillway	Spillway capacity (m^3/s)	Maximum water level (m)
Double-curvature concrete	200	380	3000 million	Gated ski-chute	16500	530

In this study, to investigate flow characteristics over the whole spillway, the CFD software, flow-3D, has been used. Moreover, the ability of this software has been examined with regard to a spillway configuration by comparing the results with experimental results.

2. Experimental model

The Shahid Abbaspour dam is a large arch dam on the Karun River located 50 kilometers northeast of Masjed Soleiman, in Khuzesten province, Iran. Some features of this dam are presented in Table 1.

The chute spillway consists of three bays with a width of 18.5 m, which are controlled by radial gates, 20×15 m in dimension [8]. The hydraulic model of this spillway was built in the hydraulic laboratory of the Iran Water Research Institute (WRI) in 1984, with a scale of 1:62.5. The spillway hydraulic model consists of part of the dam, spillway configuration, and radial gates; 250 meters from the lake upstream and 800 meters from the river downstream [9]. Based on several measurements, values of water depth, velocity and piezometric pressure were calculated along the spillway for different water levels.

3. Numerical methodology

Flow-3D is a powerful CFD software, capable of solving a wide range of fluid flow problems. It uses the finite volume method to solve RANS (Reynolds Average Navier-Stokes) equations. This software utilizes a true Volume Of Fluid method (VOF method) for computing free surface motion [10], and complex geometric regions are modeled using the area/volume obstacle representation (FAVOR) method [11]. In the true VOF method, a special advection technique is used that gives a sharp definition of the free surface and does not compute the dynamics in the void or air regions. The portion of volume or area occupied by the obstacle in each cell is defined at the beginning of the analysis, and the fluid fraction in each cell is also calculated. The continuity and momentum equations of the fluid fraction are formulated using the FAVOR function, and the finite volume method or a finite difference approximation is used for the discretization and solving of each equation [4].

The general governing mass continuity equation, including the VOF and FAVOR variables, can be written as:

$$V_F\frac{\partial p}{\partial t}+\frac{\partial}{\partial x}(\rho u A_x)+R\frac{\partial}{\partial y}(\rho v A_y)+\frac{\partial}{\partial z}(\rho w A_z)+\xi\frac{\rho u A_x}{x}=R_{\mathrm{DIF}}+R_{\mathrm{SOR}}, \tag{1}$$

where V_F is the fractional volume open to flow, ρ is the fluid density, R_{DIF} is a turbulent diffusion term, and R_{SOR} is a mass source. The velocity components (u,v,w) can be in the coordinate directions (x,y,z) or (r,θ,z). A_x, A_y and A_z are the fractional areas open to flow in $x-, y-$ and $z-$ directions, respectively.

The first term on the right side of Eq. (1) is a turbulent diffusion term:

$$R_{\mathrm{DIF}}=\frac{\partial}{\partial x}(\upsilon_\rho A_x\frac{\partial p}{\partial x})+R\frac{\partial}{\partial y}(\upsilon_\rho A_y R\frac{\partial p}{\partial y})+\frac{\partial}{\partial z}(\upsilon_\rho A_z\frac{\partial p}{\partial z})+\xi\frac{\rho\upsilon_\rho A_x}{x}, \tag{2}$$

where the coefficient $\upsilon_\rho=C_\rho\mu/\rho$ in which μ is the coefficient of momentum diffusion, and C_p is a constant whose reciprocal is usually referred to as the turbulent Schmidt number. Finally, the last term, R_{SOR}, on the right side of Eq. (1), is a density source term that can be used, for example, to model mass injection through porous obstacle surfaces [12].

In compressible flow problems, solution of the full density transport equation is required, as stated in Eq. (1). For incompressible flow problems, ρ is a constant and Eq. (1) reduces to the incompressibility condition:

$$\frac{\partial}{\partial x}(uA_x)+R\frac{\partial}{\partial y}(vA_y)+\frac{\partial}{\partial z}(wA_z)+\xi\frac{uA_x}{x}=\frac{R_{\mathrm{SOR}}}{\rho}. \tag{3}$$

The equations of motion for the fluid velocity components (u,v,w) in the three coordinate directions (the Navier-Stokes equations) can be written as:

$$\frac{\partial u}{\partial t}+\frac{1}{V_F}\left\{uA_x\frac{\partial u}{\partial x}+vA_yR\frac{\partial u}{\partial y}+wA_z\frac{\partial u}{\partial z}\right\}-\xi\frac{A_y v^2}{xV_F}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+G_x+f_x-b_x-\frac{R_{\mathrm{SOR}}}{\rho V_F}u,$$

$$\frac{\partial v}{\partial t}+\frac{1}{V_F}\left\{uA_x\frac{\partial v}{\partial x}+vA_yR\frac{\partial v}{\partial y}+wA_z\frac{\partial v}{\partial z}\right\}$$

$$+\xi\frac{A_yuv}{xV_F}=-\frac{1}{\rho}R\frac{\partial p}{\partial y}+G_y+f_y-b_y-\frac{R_{\mathrm{SOR}}}{\rho V_F}v,$$

$$\frac{\partial w}{\partial t}+\frac{1}{V_F}\left\{uA_x\frac{\partial w}{\partial x}+vA_yR\frac{\partial w}{\partial y}+wA_z\frac{\partial w}{\partial z}\right\}$$

$$=-\frac{1}{\rho}\frac{\partial p}{\partial z}+G_z+f_z-b_z-\frac{R_{\mathrm{SOR}}}{\rho V_F}w, \qquad (4)$$

where (G_x, G_y, G_z) are body accelerations, (f_x, f_y, f_z) are viscous accelerations, (b_x, b_y, b_z) are flow losses in porous medium, and the final terms account for the injection of mass at a source represented by geometry components. The term $U_w = (u_w, v_w, w_w)$ is the velocity of the source component, and the term $U_s = (u_s, v_s, w_s)$ is the velocity of the fluid at the surface of the source relative to the source itself [12].

As mentioned before, flow-3D numerically solves the governing equations using finite difference or finite volume approximations. The flow region is subdivided into rectangular cells. With each cell, there are associated local average values of dependent variables. The basic numerical method used in this software has a formal accuracy, using first order finite difference approximations with respect to time and space increments. Special precautions have been taken to maintain this degree of accuracy, even when the finite difference mesh is not uniform. Moreover, second order accurate options are also available. In any case, boundary conditions are at least first order accurate in all circumstances [12].

For example, a generic form for the finite difference approximation of the momentum equation (Eq. (4)) can be written as:

$$u_{i,j,k}^{n+1}=u_{i,j,k}^{n}+\delta t^{n+1}[-\frac{P_{i+1,j,k}^{n=1}-P_{i,j,k}^{n=1}}{(\rho\delta x)_{i+\frac{1}{2},j,k}^{n}}$$

$$+G_x\text{-FUX -FUY -FUZ +VISX-BX -WSX}],$$

$$v_{i,j,k}^{n+1}=v_{i,j,k}^{n}+\delta t^{n+1}[-\frac{P_{i,j+1,k}^{n=1}-P_{i,j,k}^{n=1}}{(\rho\delta y)_{i+\frac{1}{2},j,k}^{n}}.R_{i+\frac{1}{2}}$$

$$+G_y\text{ -FVX -FVY -FVZ+VISY -BY -WSY}],$$

$$w_{i,j,k}^{n+1}=w_{i,j,k}^{n}+\delta t^{n+1}[-\frac{P_{i,j,k+1}^{n=1}-P_{i,j,k}^{n=1}}{(\rho\delta x)_{i+\frac{1}{2},j,k}^{n}}$$

$$+G_z\text{ -FWX-FWY-FWZ +VISZ-BZ-WSZ}], \qquad (5)$$

where $R_{i+1/2}$ is related to coordinate systems and equals 1 in Cartesian coordinates. The advective, viscous and acceleration terms have an obvious meaning. For example, FUX means the advective flux of u in the x direction; VISX is the x component viscous acceleration; BX is the flow losses for a baffle normal to the x direction; WSX is the viscous wall acceleration in the x-direction, and GX includes gravitational, rotational and general non-inertial accelerations.

4. Numerical model implementation

Flow over the Shahid Abbaspour dam spillway was modeled using the flow-3D software. The geometry of the spillway was created by Auto-Cad software and exported as a stereolithographic (stl) format. Then, the stl file was directly imported into the flow-3D.

The computational domain includes 150 meters before the spillway crest, the whole spillway structure and 300 meters after the flip-bucket. Moreover, about 50 meters above the spillway crest have been considered in the computational domain in a vertical direction. The renormalization group (RNG) turbulence model has been used for simulation. The decision was made based on comments in the flow-3D user's manual [12], that the RNG turbulence model is the most accurate model available in flow-3D software.

Setting the appropriate boundary conditions has a significant effect on whether the numerical results are reflecting the actual situation. As the flow domain in this software is defined as a hexahedral in Cartesian coordinates, there are six different boundaries to be fixed. In this case, flow data from free surface flow are desired. Therefore, in a vertical direction, the top boundary was set as atmospheric pressure, and the bottom boundary was specified as the wall. Since the purpose of these simulations is to model flow rate over a spillway with different water head levels for comparison with physical model data, the upstream boundary was set as the specified pressure based on total fluid height over the spillway crest, while the downstream boundary was set as outflow. It is to be mentioned that there are also several other boundary options available in this software that could be applied to the downstream side. The boundary condition in the y-direction or the direction perpendicular to the flow is specified as the wall on both sides. Figure 1 shows the boundary conditions set at each direction.

Implementing accurate initial conditions as closely as possible to the actual flow field has a very important effect on simulation times [2]. Rectangular regions were specified at the upstream and downstream of the spillway for the initial condition, and the pressure considered as hydrostatic distribution in the vertical direction. Rectangular fluid regions were specified on the downstream and upstream sides of the spillway at the same level as the specified fluid height at boundaries. Figure 2 shows the situation of the spillway at

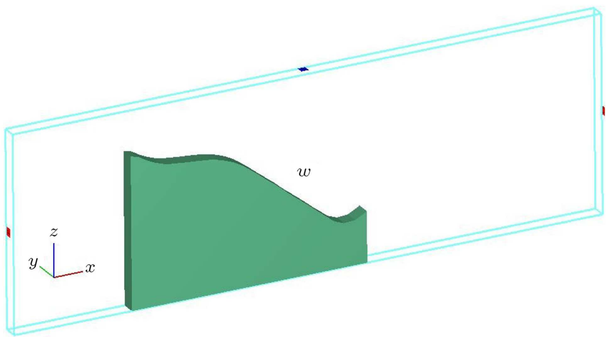

Figure 1. The boundary conditions set at each direction.

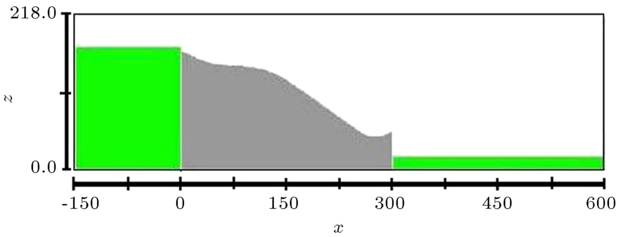

Figure 2. The situation of the spillway at the beginning of the analysis.

the beginning of the analysis. In this case, the specified fluid height was considered 181.2 m, to provide a flow rate of 1370 m^3/s.

Typically, every numerical model starts with a computational mesh, or grid. It consists of a number of interconnected cells, which subdivide the physical space into small volumes with several nodes associated with each such volume. The nodes are used to store values of the unknown parameters, such as pressure and velocity. Determining the appropriate grid size is also an important part of any numerical simulation. Grid size can affect not only the accuracy of results, but also simulation time. Therefore, it is important to minimize the number of grids while including enough resolution to sufficiently acquire significant features of geometry and the flow details. In this study, four different mesh types were considered. The size of grids and the corresponding calculation time for each mesh type are presented in Table 2. The accuracy of each mesh type compared with experimental results is shown in Figure 3. The results have been presented based on average flow depth in the discharge of 1370 m^3/s.

According to the accuracy of computed results and corresponding simulation time for each mesh type, mesh type two has been selected for the rest of the simulation, and numerical results have been presented based on this mesh size.

Table 2. The mesh type and the corresponding calculation time.

Mesh type	**Mesh size ($\mathbf{x \times y \times z}$) (m)**	**Calculation time (min)**
1	4×2.05 × 2	290
2	2 × 2.05 × 1	900
3	2 × 1.028 × 1	1220
4	2 × 2.05 × 2	510

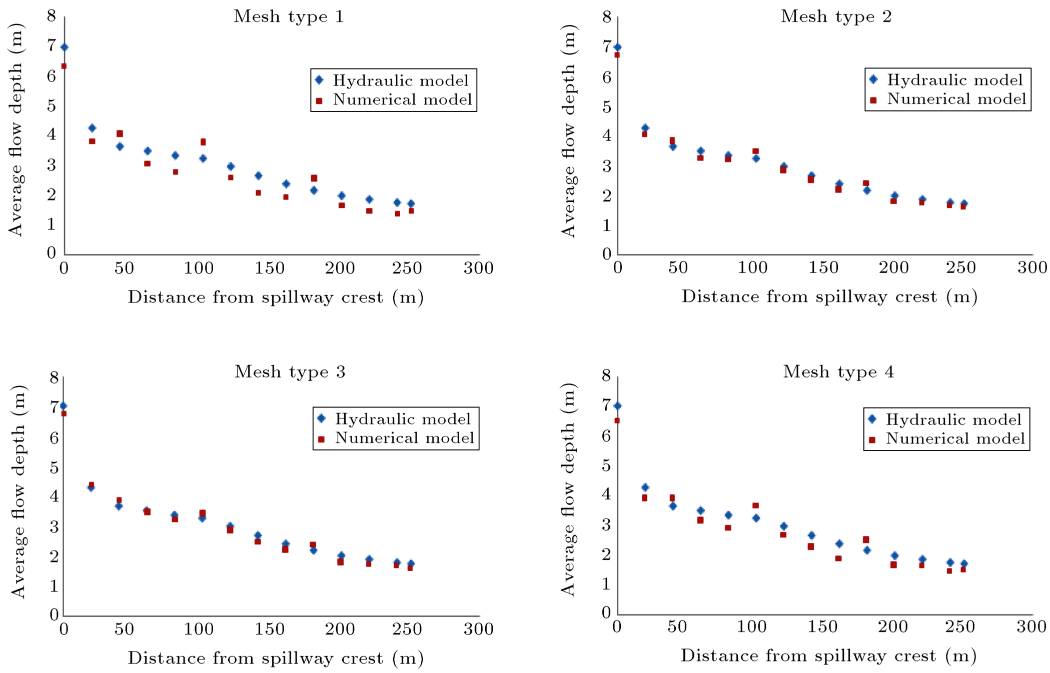

Figure 3. The accuracy of each mesh type based on average flow depth.

5. Results

By three-dimensional numerical simulation, flow characteristics such as flow depth, velocity and pressure were obtained under the condition of four different flow rates. Results have been compared with the experimental results of the hydraulic model.

The piezometric pressure values along the spillway were calculated for four different flow rates. Figures 4 to 7 show the pressure profiles along the spillway. It can be seen that there is reasonably good agreement between the results of the numerical model and those from experiments. For all flow rates, the minimum pressure on the chute occurs at a distance of 140 meters from the spillway crest. The pressure drop at this section of the chute may cause a cavitation phenomenon that can be harmful for the structure. The difference percentage between computed and measured piezometric pressure values is presented in Table 3. It can be seen that the maximum difference is 7.97%, and the values match well.

For all flow rates, the average velocity profiles calculated by the numerical model and measured by the hydraulic model are shown in Figures 8 to 11. According to the results, we discover that as the

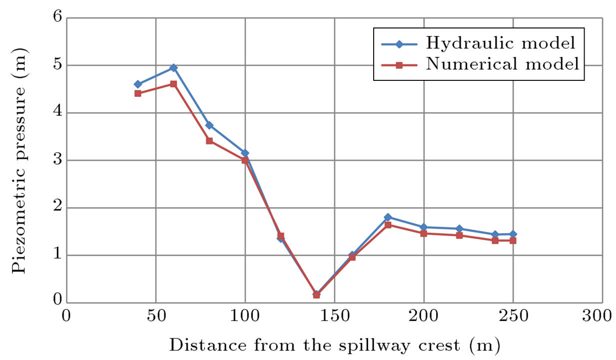

Figure 4. Comparison of computed and measured piezometric pressure for $Q = 1370$ m^3/s.

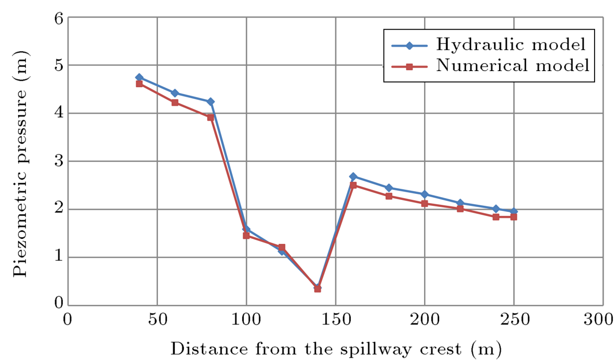

Figure 5. Comparison of computed and measured piezometric pressure for $Q = 2000$ m^3/s.

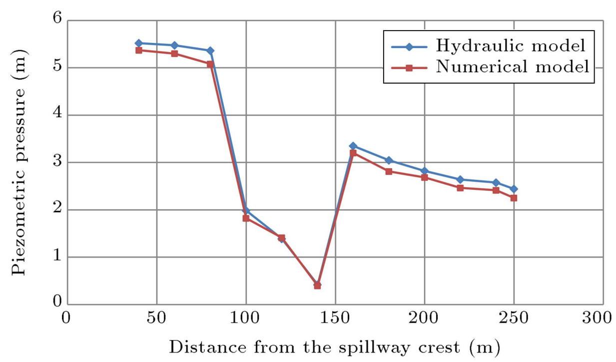

Figure 6. Comparison of computed and measured piezometric pressure for $Q = 2500$ m^3/s.

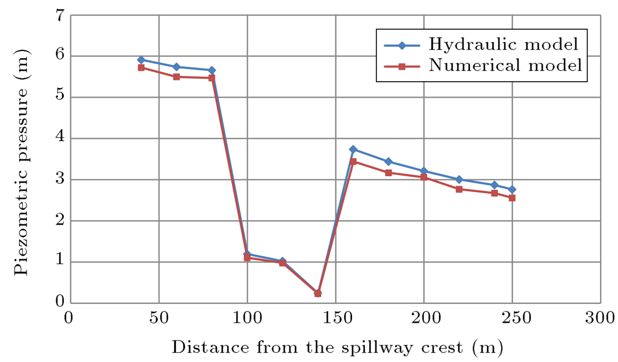

Figure 7. Comparison of computed and measured piezometric pressure for $Q = 3000$ m^3/s.

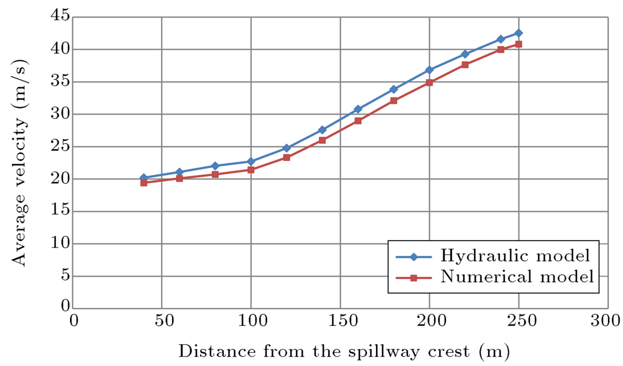

Figure 8. Comparison of computed and measured average flow velocity for $Q = 1370$ m^3/s.

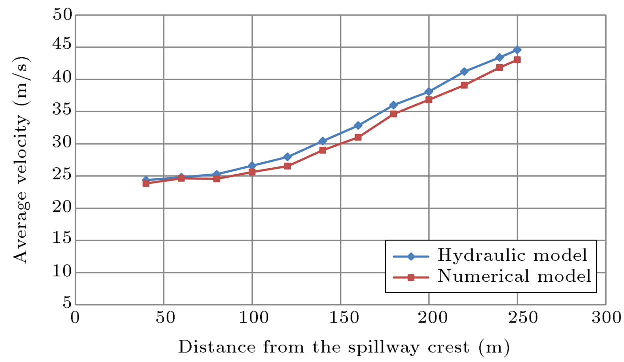

Figure 9. Comparison of computed and measured average flow velocity for $Q = 2000$ m^3/s.

Table 3. Difference between computed and measured piezometric pressure (%).

Distance from spillway crest (m)	Difference (%)			
	$Q = 1370$ m^3/s	$Q = 2000$ m^3/s	$Q = 2500$ m^3/s	$Q = 3000$ m^3/s
40	4.19	2.8	2.72	3.18
60	6.68	4.46	3.18	4.2
80	7.39	7.76	5.26	3.3
100	4.88	7.82	7.94	7.47
120	4.09	7.56	1.66	4.5
140	7.89	6.83	5.27	4.15
160	4.67	6.75	4.42	7.94
180	7.96	7.07	7.74	7.88
200	7.29	7.79	4.85	4.7
220	7.93	5.63	6.74	7.92
240	7.97	7.94	6.29	6.86
250	7.89	5.64	7.82	7.46

Table 4. Difference between computed and measured flow velocity (%).

Distance from spillway crest (m)	Difference (%)			
	$Q = 1370$ m^3/s	$Q = 2000$ m^3/s	$Q = 2500$ m^3/s	$Q = 3000$ m^3/s
40	3.96	2.14	1.34	0.35
60	4.68	0.66	1.34	2.09
80	5.37	2.8	3.27	2.68
100	5.46	3.71	5.42	5.37
120	5.27	5.11	5.16	4.59
140	5.32	4.71	4.94	4.96
160	5.26	5.46	5.47	3.84
180	5.12	3.79	3.3	4.39
200	5.28	3.31	4.37	4.61
220	4.17	5.14	4.08	3.8
240	3.84	3.61	4.14	3.91
250	4.03	3.43	3.39	3.72

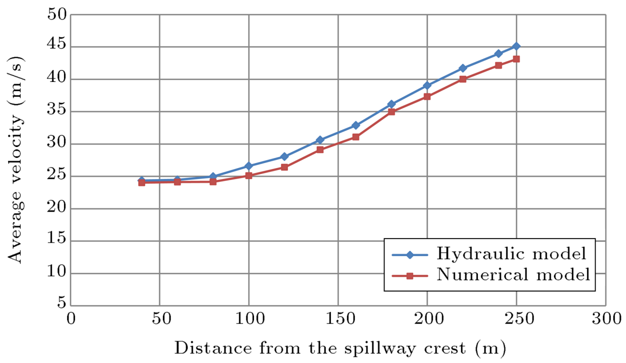

Figure 10. Comparison of computed and measured average flow velocity for $Q = 2500$ m^3/s.

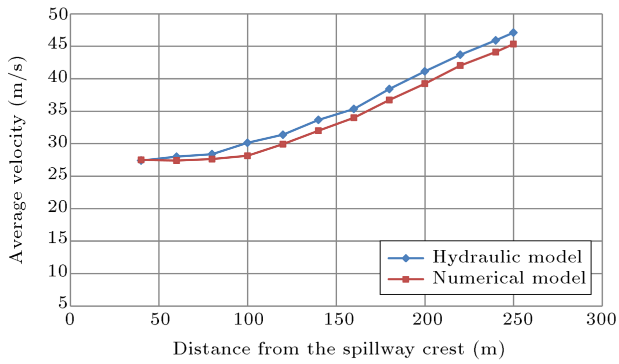

Figure 11. Comparison of computed and measured average flow velocity for $Q = 3000$ m^3/s.

flow rate increases, the flow velocity increases also. Moreover, because of the high flow velocity at the ending areas of the chute, these areas might be at risk of serious to major cavitation damage [13]. The difference percentage between calculated and experimental results is presented in Table 4. It can be seen that the maximum difference between results is 5.47%, which is less than 6% [7], and the calculated values match well with experimental results.

6. Conclusion

As physical measurements are expensive and time consuming, numerical modeling can be a convenient and effective tool for analyzing flow over spillways. Using numerical modeling can provide detailed information of the complete flow field and ensure the correctness of the design.

In this study, flow over a chute spillway was simulated using CFD software, flow-3D. It was seen that the numerical results agreed well with experiments. The maximum difference between calculated and experimental results in average velocity values was 5.47% and, in piezometric pressure values, was 7.97%. Evaluation of the capability of this software for numerical simulation of flow over a spillway proved to be successful.

References

1. Khatsuria, R.M. "Hydraulics of spillways and energy dissipators", 1st Ed., pp. 2-15, Marcel Dekker, New York, USA (2005).
2. Chanel, P.G. "An evaluation of computational fluid dynamics for spillways", Thesis (MSC), University of Manitoba, Canada, pp. 11-23 (2008).
3. Savage, B.M. and Johnson, M.C. "Flow over ogee spillway: Physical and numerical model case study", *J. of Hydraul. Eng., ASCE*, **127**(8), pp. 640–649 (2001).
4. Ho, H., Boyes, K., Donohoo, S. and Cooper, B. "Numerical flow analysis for spillways", *Proc., 43rd ANCOLD Conf.*, Hobart, Tasmania, pp. 24-29 (2003).
5. Kim, D.G., and Park, J.H. "Analysis of flow structure over ogee spillway in consideration of scale and roughness effect by using CFD model", *J. of Civil Eng., KSCE*, **9**(2), pp. 161-169 (2005).
6. Dargahi, B. "Experimental study and 3D numerical simulations for a free-overflow spillway", *J. of Hydraul. Eng., ASCE*, **132**(9), pp. 899-907 (2006).
7. Zhenwei, M., Zhiyan, Z. and Tao, Z. "Numerical simulation of 3-D flow field of spillway based on VOF method ", *Procedia Engineering*, **28**, pp. 808-812 (2012).
8. Mahab Ghodss Consulting Engineers, *Karun Model Spillway*, Hydraulic Department, Tehran (1984).
9. Water Research Institute, Iran Ministry of Energy, *Hydraulic Model of the Spillway of Shahid Abbaspour Dam,* Dept. of Hydro-Environment, Tehran, Iran (1984).
10. Hirt, C.W. and Nichols, B.D. "Volume of Fluid (VOF) method for the dynamics of free boundaries", *J. Comp. Phys.*, **39**(1), pp. 201-225 (1981).
11. Hirt, C.W. and Sicilian, J.M. "A porosity technique for the definition of obstacles in rectangular cell meshes"., *Proc. 4th Int. Conf. Ship Hydro*, National Academy of Science, Washington, DC, September, pp. 1-19 (1985).
12. Flow Science, Inc. "Flow-3D user's manuals, version 9.2", pp. 110-114, Santa Fe, NM (2007).
13. Fadaei Kermani, E., Barani, G.A., and Ghaeini-Hessaroeyeh., M. "Investigation of cavitation damage levels on spillways", *World Applied Sciences Journal*, **21**(1), pp. 73-78 (2013).

Biographies

Ehsan Fadaei Kermani received his BS degree in Civil Engineering from Shahid Bahonar University of Kerman in 2010 and his MS degree in Hydraulic Structures in 2012. His research interests include computational fluid dynamics, data mining and artificial intelligence approaches. He has published and presented papers in various journals and at national and international conferences.

Gholam Abbas Barani received his BS degree in Civil Engineering from Tehran University in 1973, and received his MS degree in 1977 from the University of California, Davis, where he continued his research work towards a PhD degree. His research interests include sediment scour in hydraulic structures, water resource engineering, optimization of reservoir operation, and hydrology. He has published more than 300 papers in various journals and presented many others at national and international conferences.