2013, Scienceline Publication

Journal of Civil Engineering and Urbanism

Volume 3, Issue 4: 150-155 (2013) (Received: January 08, 2013; Accepted: May 12, 2013; Published: July 30, 2013) ISSN-2252-0430

Determination of Energy Dissipation in Stepped Spillways Using Finite Element and Finite Volume Methods

Farinaz Shoja[1], Mohammad Reza Nikpour[2]*, and Hojjat Sadeghi[3]

[1] *M.Sc. of hydraulic structures, Department of Water engineering, Agricultural faculty, University of Tabriz, Tabriz- Iran.*
[2] *Ph.D. Candidate of hydraulic structures, Department of Water Engineering, Faculty of Agriculture, University of Tabriz, Tabriz, Iran.*
[3] *M.Sc. Student of hydraulic structures, Department of civil engineering, Islamic Azad University, Maragheh, Iran.*

*Corresponding author's Email address: nikpoor.reza@gmail.com

ORIGINAL ARTICLE

ABSTRACT: Accurate and suitable design of Spillways as major parts of dams is very important in the stability and safety of dams. In this study, flow over stepped spillway was simulated in 2D by solving poisson functions of stream and pressure in Finite Element method and with analyzing k-ε turbulence and volume of fluid models in Finite Volume technique. Also a comparison was done between the results of numerical models and measured values in order to investigate the accuracy of the models in calculation of energy dissipation. The relative error percent average of energy dissipation calculation using Finite Element and Finite Volume approaches was achieved 2.85% and 1.26%, respectively. Results showed Finite Volume approach could be employed slightly better than Finite Element method in modelling flow hydraulic over stepped spillway.

Keywords: Finite Element, Finite Volume, Stepped Spillway, Turbulence Model, Volume of Fluid Model

INTRODUCTION

Spillway is a hydraulic structure that usually is used in detention and storage dams to release extra water and flood in emergency situations. Up to now many dams are destroyed because of inefficiency and non convenient design of their spillways. Therefore, the analysis of water flow over a spillway is an important engineering problem (Chatila and Tabbara, 2004). In recent years, more attention have been paid to the stepped spillways due to their significant effect on energy dissipation and their adaptability to be built by technology of roller compacted concrete (RCC). Since the rate of energy dissipation in stepped spillways is more than ogee spillways with the same dimensions, the proper use of these spillways can reduce size of the structures, such as the reduction of drilling depth, length and height of the sidewalls of downstream stilling basin, it can also have economical benefits (Chanson, 1995). Historically, scaled physical models have been constructed in hydraulic laboratories to determine characteristics of flow hydraulic over spillways. But they are expensive and time-consuming. Today, with the advance in computer technology, numerical methods are used to simulate complex problems of fluid flow as reliable and affordable approaches. Moreover, different geometries of spillway can be simulated using numerical models and the best option in terms of hydraulic conditions can be chosen to build the main model. Due to the importance of the issue, many researchers have made several attempts to get better results. Cassidy (1965) for the first time used dimensional analysis and assumption of irrotational flow to analyze the flow over the spillway. He calculated the water surface profile and discharge coefficient of the spillway. Henderson et al. (1991) investigated the risk of cavitation on spillways with the help of Boundary Element method and found acceptable results. Savage and Johnson (2001) simulated flow over an ogee spillway using FLOW-3D software in 2D. The results of the numerical model including pressure on the spillway crest, water surface profile, and discharge coefficient of the spillway were in very good agreement with the experimental values. Hasani (2003) used potential functions by DOT software in order to analyze the flow over ogee spillway with the assumptions of incompressible, non- viscous and irrotational flow. Then, he calibrated the mentioned numerical model using measured values of an actual model. The model had acceptable yield in calculating of water surface profile and velocity values in different points of the spillway. Tabbara et al. (2005) performed some experiments on four stepped spillways with different geometries and simulated them by means of ADINA software. In all of the cases, values of water surface profile and energy dissipation obtained from the numerical model were close to those of the experimental models. Dargahi (2006) used FLUENT software for simulation of flow over ogee spillway in 3D. After examining different turbulence models, RNG model revealed the best results compared to the experimental data. Ferrari (2010) used a mesh less method called SPH[1] to analyze the flow over ogee spillway. The results of numerical model compared to physical model were in good agreement. Karimi and Mousavi Jahromi (2011) studied flow over a stepped spillway by ANSYS software which operates on Finite Element method. They reported that the accuracy of flow simulation was appropriate. In the present study, flow over a stepped spillway was simulated in 2D by solving

[1] Smooth Particle Hydrodynamics

To cite this paper: Shoja F., Nikpour M.R. and Sadeghi H. 2013. Determination of Energy Dissipation in Stepped spillways using Finite Element and Finite Volume Methods. *J. Civil Eng. Urban.*, 3 (4): 150-155.
Journal homepage: http://www.ojceu.ir/main/

poisson functions of stream and pressure in Finite Element method and with analyzing k-ε turbulence models and volume of fluid model in Finite Volume technique. Furthermore, a comparison was done between the results of numerical models and measured values in order to investigate the accuracy of the models in calculation of energy dissipation.

MATERIALS AND METHODS

Finite volume method

The Finite Volume (FV) method uses the integral form of the conservation equations as its starting point. The solution domain is subdivided into a finite number of contiguous control volumes (CVs), and the conservation equations are applied to each CV. At the centroid of each CV, lies a computational node at which the variable values are calculated. Interpolation is used to find the variable values at the CV surface in terms of the nodal values. Surface and volume integrals are approximated using suitable quadrature formula. As a result, an algebraic equation obtains for each CV, in which a number of neighbor nodal values appear. The FV method can accommodate any type of grid, so it is suitable for complex geometries (Ferziger and Peric, 2002).

Finite element method

The Finite Element (FE) method is similar to FV method in many ways. The domain is broken into a set of discrete volumes or finite elements that are generally unstructured. In 2D, they are usually triangles or quadrilaterals, while in 3D tetrahedral or hexahedral are most often used. The FE method is based on two general methods including Weighted Residual and Variational Calculating. Galerkin is the most common method of the Weighted Residual approach. The distinguish feature of FE method is that the equations are multiplied by a shape function before they are integrated over the entire domain. In the simplest FE methods, the solution is approximated by a linear shape function within each element. An important advantage of the FE method is the ability to deal with arbitrary geometries (Ferziger and Peric, 2002).

The governing equations

The governing differential equations on flow over spillways in two-dimensional conditions involve Navier – Stokes equations including a continuity equation and two momentum equations. Assuming incompressible, non- viscous and irrotational flow, the number of governing equations and the unknown parameters of the problem can be reduced by stream functions. This will lead to spending less on the numerical analysis of such flow. In the momentum equations, by deriving from u (velocity component in x direction) with respect to y parameter and also deriving from v (velocity component in y direction) with respect to x and then subtracting the equations, equation 1 will be obtained:

$$\frac{\partial}{\partial t}\left(\frac{\partial u}{\partial y}-\frac{\partial v}{\partial x}\right)+u\frac{\partial}{\partial x}\left(\frac{\partial u}{\partial y}-\frac{\partial v}{\partial x}\right) +v\frac{\partial}{\partial y}\left(\frac{\partial u}{\partial y}-\frac{\partial v}{\partial x}\right)+\frac{\partial u}{\partial x}\frac{\partial u}{\partial y}+\frac{\partial u}{\partial y}\frac{\partial v}{\partial y} -\frac{\partial u}{\partial x}\frac{\partial v}{\partial x}-\frac{\partial v}{\partial x}\frac{\partial v}{\partial y}=0 \qquad (1)$$

The fourth and fifth sentences of the formula 1 are a multiple of $\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}$ which according to the continuity equation have a value equal to zero. By replacing 2-dimensional vorticity transport equation, according to formula 2, in formula 1, equation 3 will be obtained.

$$\zeta=\frac{\partial u}{\partial y}-\frac{\partial v}{\partial x} \qquad (2)$$

$$\frac{\partial \zeta}{\partial t}+u\frac{\partial \zeta}{\partial x}+v\frac{\partial \zeta}{\partial y}=0 \qquad (3)$$

On the other hand, the stream function (Ψ) is defined as formulas 4 and 5 as following:

$$u=\frac{\partial \psi}{\partial y} \qquad (4)$$

$$v=-\frac{\partial \psi}{\partial x} \qquad (5)$$

So, the vorticity transport equation leads to poisson function of stream according to equation 6:

$$\nabla^2\psi=\zeta \qquad (6)$$

Thus, the three equations of continuity and momentum in 2-dimensional flow of incompressible fluids are changed into two equations including the vorticity transport function (ξ) and the poisson function of stream (Ψ). The components of u and v can be obtained by solving these two equations. After computing values of the stream function, distribution of pressure can also be acquired by solving poisson function of pressure. The pressure formula can be obtained by deriving from the components of u and v and momentum equations with respect to x and y, and then summing them:

$$\frac{\partial}{\partial t}\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}\right)+\left(\frac{\partial u}{\partial x}\right)^2+\left(\frac{\partial v}{\partial y}\right)^2 +2\left(\frac{\partial v}{\partial x}\right)\left(\frac{\partial u}{\partial y}\right)+u\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 v}{\partial x\partial y}\right) +v\left(\frac{\partial^2 u}{\partial x\partial y}+\frac{\partial^2 v}{\partial y^2}\right)=-\nabla^2 p \qquad (7)$$

First, fifth and sixth sentences of the equation 7 are multiples of the continuity equation and therefore have values equal to zero. Thus, the pressure equation can be summarized as follows:

$$-\nabla^2 p=\left(\frac{\partial u}{\partial x}\right)^2+\left(\frac{\partial v}{\partial y}\right)^2+2\left(\frac{\partial v}{\partial x}\right)\left(\frac{\partial u}{\partial y}\right) \qquad (8)$$

To cite this paper: Shoja F., Nikpour M.R. and Sadeghi H. 2013. Determination of Energy Dissipation in Stepped spillways using Finite Element and Finite Volume Methods. *J. Civil Eng. Urban.*, 3 (4): 150-155.
Journal homepage http://www.ojceu.ir/main/

The final form of the poisson function of pressure is obtained as equation 9:

$$\nabla^2 p = 2\left[(\frac{\partial^2\psi}{\partial x^2})(\frac{\partial^2\psi}{\partial y^2}) - (\frac{\partial^2\psi}{\partial x\partial y})^2\right] \quad (9)$$

Case study

In the present study, experimental findings of Chanson and Toombes (2001) were used in order to calibrate the numerical models. A scheme of the stepped spillway is depicted in Figure 1. Points 1 and 2 show the measuring places of velocity and depth values in order to determine rate of energy dissipation in experimental model. The geometric parameters of the investigated stepped spillway are presented in table 1.

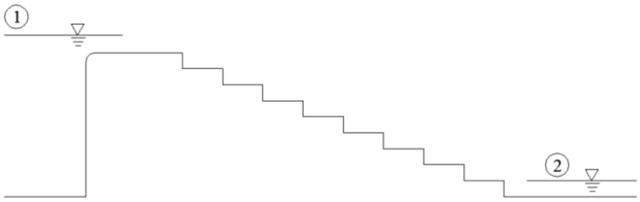

Figure 1. Experimental model scheme of Chanson and Toombes (2001).

Table 1. Geometric parameters of the investigated stepped spillway.

Parameter	Values
Width of spillway	1m
Number of steps	9
Horizontal length of steps	0.25m
Step height	0.1m
Slop angle of spillway	21.8°
Length of broad crested weir	0.6 m

Boundary conditions in the FV method

In the present study, FLUENT software was employed in order to apply the FV method. FLUENT is one of the most popular and suitable software of CFD that provides a wide range of advanced physical models for fluid flow and heat transfer including multiphase flow. It can exchange 2D and 3D dominant differential equations to algebraic equations by using the FV method. In this study, in order to simulate turbulent flow, 2D k-ε models including Standard, RNG[1] and Realizable were used and volume of fluid (VOF) model was used to simulate two-phase flow of water and air. It should be noted that the value of VOF for determining the position of free water surface is considered equal to 0.5 (Dargahi, 2006). Mesh generation and definition of boundary conditions was done by GAMBIT software. As shown in Figure 2, a 2D grid was used for mesh generation which consisted of Tri (pave) and Quad (map) elements. Furthermore, according to Figure 2, boundary conditions were defined as following: S_1: Pressure-Inlet, S_2: Wall, S_3: Pressure-Outlet and S_4: symmetry.

[1] Renormalization-group k-ε model (RNG)

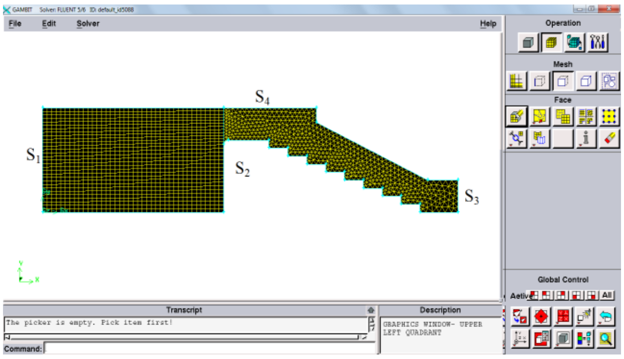

Figure 2. The meshed model with definite boundary conditions in GAMBIT

Boundary conditions in the FE method

In this study, a code ,developed by the authors of this paper, in Fortran-90 was used in order to employ the Galerkin FE method. This code was set based on numerical solution of the poisson function of stream (Eq. 6) and pressure (Eq. 9) in the Galerkin method. The process of the application of this code was as following: first, the position of water free surface was assumed and then the area was meshed by triangular elements. Figure 3 shows a sample of grid generated by the code. It should be noted that this code allows users to change distances and number of elements in desired points. Since in the first phase of applying the code the poisson function of stream (Eq. 6) was employed, the boundary conditions were defined on basis of the stream function. The formulas 10 to 13 show the boundary conditions defined in the code in accordance with the Figure 3.

$$S_1: \quad \frac{\partial\psi}{\partial n} = 0 \quad (10)$$

$$S_2: \quad \frac{\partial\psi}{\partial n} = 0 \quad (11)$$

$$S_3: \quad \psi = 0 \quad (12)$$

$$S_4: \quad \psi = q(m^3/s/m) \quad (13)$$

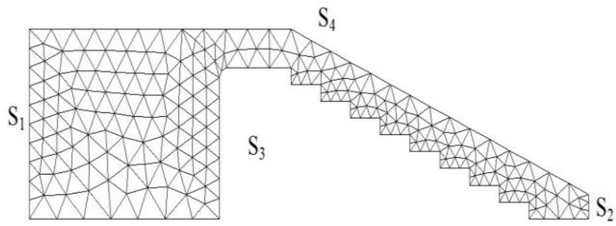

Figure 3. The meshed model by the code in Fortran-90 with definite boundary conditions.

According to the Figure 3, in S1 and S2 which they are the upstream and downstream boundaries, respectively, velocity distribution was assumed to be uniform. Pressure distribution was also considered as hydrostatic. Moreover, S3 and S4 which are the solid and flow free surface boundaries, respectively, were considered as stream line. After applying the boundary conditions and solving the poisson function of stream, the values of u and v were computed and velocity magnitude was obtained from formula 14.

$$V = \sqrt{u^2 + v^2} \quad (14)$$

Besides, for the S_4 bound, other than the considered boundary condition (formula 13), one more boundary condition should be provided which is the relative pressure equal to zero on free surface. However, satisfying this condition was not possible due to the approximation of the free surface at the beginning of the solution process. Thus, this condition was established using more iterations (trial and error) and by approaching to the real free surface. In the next phase, the poisson function of pressure was solved in order to compute the values of pressure in the elements. By investigating the vertical component of velocity and the relative pressure on the free surface, providing that the assumed position of water free surface was inaccurate, its position was changed vertically and the poisson functions of stream and pressure were solved once again. This operation was continued until the global convergence was achieved and both of the boundary conditions were established on the free surface.
After performing the numerical models, the obtained values of depth and velocity for 9 different flow rates were extracted by the models in the points 1 and 2 (Figure 1). Then, the rate of relative energy dissipation was calculated by formula 15.

$$\Delta E\,(\%) = \frac{E_1 - E_2}{E_1} \times 100 \qquad (15)$$

In the above formula, E_1 and E_2 refer to the values of specific energy in points 1 and 2. Finally, in order to examine the efficiency of the numerical models, the parameter of relative error percentage for calculated energy dissipation compared to the measured values was separately calculated by formula 16.

$$RE\,(\%) = \left| \frac{measured - calculated}{measured} \right| \times 100 \qquad (16)$$

RESULTS AND DISCUSSION

In order to find the lowest required elements which had the least computational error for the numerical model, number of elements were successively increased in some different ranges for constant boundary conditions. Figures 4 and 5 show variations of computation error versus the number of elements in calculating the energy dissipation for the FE and FV methods, respectively. As it is shown in the Figure 4, in the FE method, the value of relative error is not significantly reduced by increasing the number of elements to more than 3283. In other words, for these numbers of the elements, the numerical model operates independently from the grid. In addition, regarding the Figure 5, the FV method operates independently from the grid for 8754 elements. On the other hand, considering that increasing the number of elements increases the necessary time for performing computations, therefore, the arrangement of the grids was designed so that depending on the sensitivity of the solution areas, elements with different sizes were used.

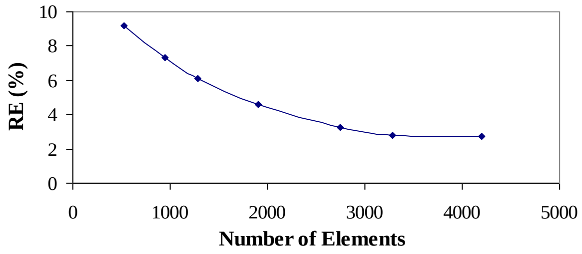

Figure 4. Variations of computation error versus the number of meshesin the FE method.

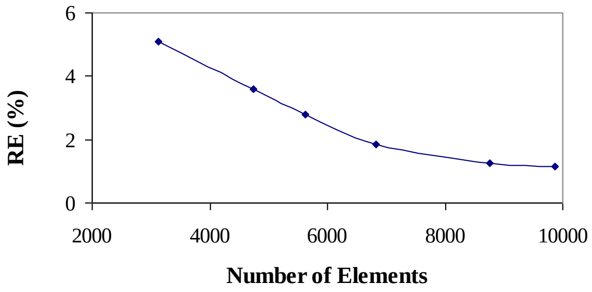

Figure 5. Variations of computation error versus the number of meshesin the FV method.

Figure 6 shows the overflow of water over the stepped spillway for flow rate equal to 0.158 (m^3/s) in FLUENT software.

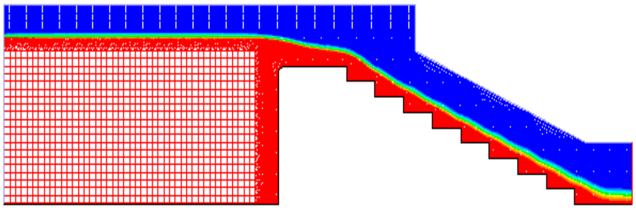

Figure 6. Flow passing over the stepped spillway for Q = 0.158 m^3/s in FLUENT.

Figure 7 shows the variations of energy dissipation of the spillway for the measured and computational values. As it seen, the rate of energy dissipation was reduced by increasing the passing flow over the spillway. In fact, by increasing the amount of flow rate, the depth and velocity values of the flow in the downstream of the spillway (point 2) were increased and it led to the reduction of the energy loss of the flow. On the other hand, for all the values of flow rates, the energy dissipation calculated by the FE method was less than the measured values. This is because of not considering the dissipation of kinetic energy in poisson function of stream and calculating the value of velocity in point 2 higher than its real value. In other words, the stream function operates relatively weak in the analysis of the areas which have high turbulence intensity. As it mentioned, in the FV method in order to analyze turbulent flow, k-ε models including Standard, RNG and Realizable were used. After extracting the results and comparing them with the measured values, the RNG model performed better than two other models because it can well simulate rotational flows. As a result, all of the mentioned values in the FV method are the results obtained from the application of the RNG model.

To cite this paper: Shoja F., Nikpour M.R. and Sadeghi H. 2013. Determination of Energy Dissipation in Stepped spillways using Finite Element and Finite Volume Methods. *J. Civil Eng. Urban.*, 3 (4): 150-155.
Journal homepage: http://www.ojceu.ir/main/

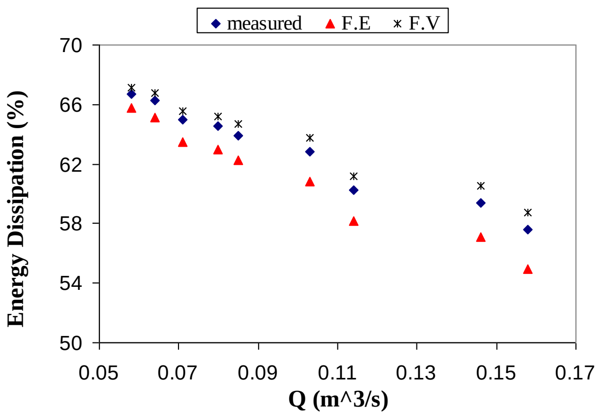

Figure 7. Variations of energy dissipation of the spillway for the measured and computational values.

The relative error (RE) of the numerical models in estimation of the relative energy dissipation (archived by formula16) is presented in Table 2. According to this table, values of RE in the both models is increased by increasing the flow rate. In fact, the increase of flow rate has caused the increase of turbulence intensity in the steps and this has led to the increase in computation error of the numerical models. In this regard, values of RE in the FE method are more than the FV method. As it was already mentioned, in the FV method, the values of velocity were obtained by solving the turbulence equations while, in the FE method, these values were obtained by solving the poisson equations of stream. In order to obtain the poisson functions of stream and pressure, it was assumed that the flow was irrotational, moreover, for the upstream and downstream boundaries (S1 and S2 in Figure 3) the stream lines were assumed to be parallel and the pressure distribution was assumed to be hydrostatic. While the stream lines on the crest and downstream of the spillway are not parallel and are very much curved which causes the creation of acceleration magnitude and centrifugal force that is vertical to the flow direction. Therefore, the assumptions applied in the potential flows were incorrect for the steps and had led to increase of the RE values in the FE method in calculating the velocity.

Table 2. RE values of the numerical models in estimation of the energy dissipation.

Q (m^3/s)	F.E Method	F.V Method
0.058	0.67	1.36
0.064	0.74	1.67
0.071	0.82	2.37
0.08	1.07	2.45
0.085	1.24	2.58
0.103	1.48	3.20
0.114	1.56	3.47
0.146	1.85	3.94
0.158	1.93	4.62
Average	1.26	2.85

CONCLUTION

In the present study, the FE and FV methods were employed for 2D simulation of the flow over a stepped spillway. FLUENT software was used to employ the FV method. Also, in order to apply the Galerkein FE method, a code developed in Fortran-90 was employed. In FLUENT, the RNG model was used to analyze the turbulence and the VOF model was used to simulate the two-phase flow. The code was based on the discretization and solving the poisson equations of stream and pressure through the Galerkin method. After applying the numerical models for 9 different flow rates, the parameters of depth and velocity in upstream and downstream of the spillway were extracted. Based on these parameters, the rate of energy dissipation was calculated and compared with the experimental values. The average of relative error percentage in calculating energy dissipation for the FE and FV methods were %2.85 and %1.26, respectively. The findings of the study revealed that the both numerical models had acceptable performance in simulation of the flow. On the other hand, the results indicated that the intensity of flow turbulence increased by increasing the flow rate and it led to reduction of the numerical models accuracy in hydraulic simulation. Generally, the calculation error of the FE method was more than the FV method. Among the influential factors, inaccuracy of the applied assumptions in potential flows on the crest and downstream of the spillway can be mentioned. Therefore, it can be inferred that in order to simulate the parts which are curved and in which the flow turbulence is very high, employing turbulence models can yield better performance.

REFRENCES

Cassidy J.J. (1965). Irrotational flow over spillways of finite height. Journal of Engineering Mechanics Division, 91(6):155-173.

Chanson H, Toombes L. (2001). Experimental investigations of air entrainment in transition and skimming flows down a stepped chute. Application to embankment overflow stepped spillways. Research Report No. CE158, Department of Civil Engineering, University of Queensland, Brisbane, Australia. 77pp.

Chanson H. (1995). Hydraulic Design of Stepped Cascades, Channels, Weirs and Spillways. Pergamon, Oxford, UK, 292 pp.

Chatila J, Tabbara M. (2004). Computational modeling of flow over an ogee spillway. Journal of computers and structures, 82: 1805-1812.

Dargahi B. (2006). Experimental study and 3D numerical simulations for a free over flow spillway. Journal of Hydraulic Engineering, ASCE, 132(9): 899-907.

Ferrari A. (2010). SPH simulation of free surface flow over a sharp-crested weir. Advances in Water Resources, 33: 270-276.

Ferziger J.H, Peric M. (2002). Computational methods for fluid dynamics. Third edition, Springer publisher. 374 pp.

Hasani A. (2003). Analysis of potential flow over a spillway by mathematical optimization. M.Sc.

Thesis, University of Science and Technology, Tehran, Iran.

Henderson H.C, Kok M, Dekoning W.L. (1991). Computer-aided spillway design using the boundary element method and non-linear programming. International journal of numerical methods in Fluids, 13: 625-641.

Karimi M, Mousavi Jahromi H. (2011). Recognizing the regime of flow over stepped spillway by ANSYS. Proceedings of the 9th hydraulic Conference, University of Tarbiat Modares, Tehran, Iran.

Savage B.M, Johnson C. (2001). Flow over ogee spillway: Physical and numerical model case study. Journal of Hydraulic Engineering, ASCE, 127(8): 640-649.

Tabbara M, Chatila J, Awwad R. (2005). Computational simulation of flow over stepped spillways. Journal of computers and structures, 83: 2215-2224.