Available online at www.CivileJournal.org

Civil Engineering Journal

Vol. 2, No. 9, September, 2016

Numerical Detection of Cavitation Damage on Dam Spillway

E. Fadaei-Kermani [a*], G. A. Barani [a], M. Ghaeini-Hessaroeyeh [a]

[a] *Department of Civil Engineering, Faculty of Engineering, Shahid Bahonar University of Kerman, Kerman, Iran*.

Received 10 August 2016; Accepted 22 September 2016

Abstract

The present paper deals with the numerical detection of cavitation damage level and location on dam spillways. At first, flow over a spillway was simulated using the computational fluid dynamics method. The flow characteristics such as pressure, velocity and depth through the spillway have been calculated for five different flow rates. Since the actual flow is turbulent, the RNG turbulence model has been used for simulation. The numerical results of flow characteristics including flow depth, velocity and pressure were compared with the available results of the hydraulic model tests. The numerical results agreed well with the experimental data, and reasonable values for the normalized root mean square error (NRMSE= 0.0476) and coefficient of determination (r^2=0.8354) indicated that the numerical model is accurate. Finally occurrence of cavitation damage to the Doosti dam spillway was investigated. Based on cavitation index, five different damage levels from no damage to major damage have been considered. Results showed that the spillway may be at the risk of cavitation damage, and the serious damage can occur at ending parts of the structure.

Keywords: Spillway; Cavitation Damage; Cavitation Index; Numerical Modeling.

1. Introduction

Spillways are important hydraulic structures that play a remarkable role in safety and stability of dams. They are designed to prevent overtopping of dams and provide sufficient safety and stability during floods. High flow velocities on dam spillways could lead to low pressure and create cavitation. Cavitation is one of the most complex phenomenons affected on the spillway, and it causes damage on the structure over time [1, 2]. Occurrence of cavitation can be a function of pressure and velocity, boundary roughness, operation duration, the amount of dissolved air in the water and strength of materials from which the boundary is constructed [3].

For a long time, problems related to cavitation phenomenon on hydraulic structures especially dam spillways, have been one of the important engineering challenges all around the world, and various efforts have been made to investigate the mechanism of cavitation damage on dam spillways. Experimental modelling such as Hay [4], Nie [5], Dong and Su [6] and Frizell et al. [7], has successfully investigated cavitation damage on spillways under controlled laboratory conditions. Moreover, using numerical modeling can be a powerful tool for investigation of flow characteristics, for example, Savage and Johnson [8], Qian et al [9] and Zhenwei et al [10].

Recent development in computer science and numerical techniques has advanced the use of computational fluid dynamics (CFD) as a powerful tool for flow simulation over spillways. Computational fluid dynamics (CFD) is a numerical method used to solve problems involving fluid flow. It can provide a faster and more economical solution than physical models; therefore engineers are interested in verifying the capability of CFD softwares [11].

In this study, to investigate cavitation damage on the Doosti Dam spillway located in Iran, the Flow-3D software has been used to simulate flow over the spillway. Flow characteristics over the whole spillway have been determined, and the accuracy of the results has been examined by making comparison of the numerical model and available experimental tests. Finally, the cavitation damage levels and locations were predicted along the spillway.

* Corresponding author: ehsanhard@gmail.com

2. Numerical Model

The Doosti dam (Iran–Turkmenistan Friendship Dam) is an earth-fill dam on the Hariroud River, which forms part of the international boundary between Iran and Turkmenistan. The features of this dam are presented in Table 1.

The dam ogee chute spillway consists of eight gated bays with a width of 15 m. The hydraulic model of this spillway has been built in hydraulic laboratory of Iran Water Research Institute (IWRI) with scale of 1:50. The spillway hydraulic model consists of a part of the dam, the spillway configuration, stilling basins, gates and a part of upstream lake and downstream river (Figure 1).

According to various measurements, values of water depth, velocity and piezometric pressure were calculated at 13 stations along the spillway for different water levels. The location of the stations is presented in Table 2.

Table 1. Some features of the Doosti dam [16]

Type	Height (m)	Length (m)	Crest Width (m)	Reservoir capacity (m^3)	Type of spillway	Active capacity (m^3)	Energy dissipater type
Earth-fill with silt-clay core	78	655	15	1250 million	Ogee gated chute	735 million	settling basin

Table 2. Situation of measurement stations

Station	1	2	3	4	5	6	7	8	9	10	11	12	13
Distance from spillway crest	0	7.27	21.44	49.98	86.14	126.97	185.36	220.44	267.98	302.78	350.84	396.41	436.4
Elevation (m)	470	466.5	464.5	462.5	459.5	448.5	432.5	423.5	417.5	413.5	408	404.5	401.5

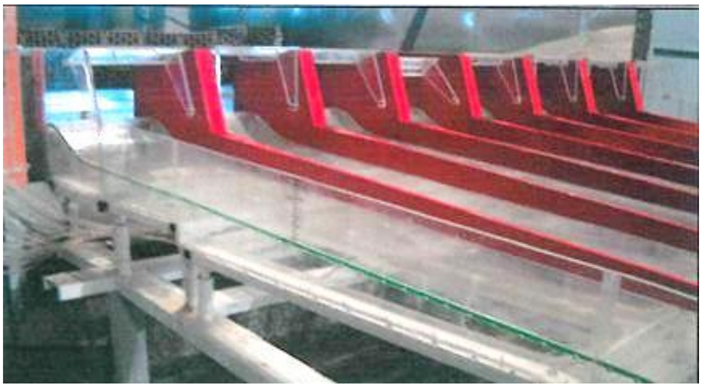

Figure 1. Doosti dam spillway hydraulic model

Flow-3D is a CFD software, capable of solving fluid flow problems by using the Finite Volume Method (FVM) to solve RANS (Reynolds Average Navier-Stokes) equations. The software utilizes a true Volume of Fluid (VOF) method to compute free surface motion, and complex geometric regions are modeled by the Fractional Area/Volume Obstacle Representation (FAVOR) method [12]. Using FAVOR function, the continuity and momentum equations of fluid can be formulated, and the FVM or a finite difference approximation is used for discretization and solving the governing equations [13].

The general governing equations consist of mass equation and the equations of motion for the fluid velocity components (u, v, w) in the three coordinate directions, including the VOF and FAVOR variables can be written as:

$$V_F \frac{\partial p}{\partial t} + \frac{\partial}{\partial x}(\rho u A_x) + R\frac{\partial}{\partial y}(\rho v A_y) + \frac{\partial}{\partial z}(\rho w A_z) + \xi \frac{\rho u A_x}{x} = RDIF + RSOR \quad (1)$$

$$\begin{aligned}
&\frac{\partial u}{\partial t} + \frac{1}{V_F}\left\{ uA_x \frac{\partial u}{\partial x} + vA_y R\frac{\partial u}{\partial y} + wA_z \frac{\partial u}{\partial z} \right\} - \xi \frac{A_y v^2}{xV_F} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + G_x + f_x - b_x - \frac{RSOR}{\rho V_F} u \\
&\frac{\partial v}{\partial t} + \frac{1}{V_F}\left\{ uA_x \frac{\partial v}{\partial x} + vA_y R\frac{\partial v}{\partial y} + wA_z \frac{\partial v}{\partial z} \right\} + \xi \frac{A_y uv}{xV_F} = -\frac{1}{\rho} R\frac{\partial p}{\partial y} + G_y + f_y - b_y - \frac{RSOR}{\rho V_F} v \\
&\frac{\partial w}{\partial t} + \frac{1}{V_F}\left\{ uA_x \frac{\partial w}{\partial x} + vA_y R\frac{\partial w}{\partial y} + wA_z \frac{\partial w}{\partial z} \right\} = -\frac{1}{\rho}\frac{\partial p}{\partial z} + G_z + f_z - b_z - \frac{RSOR}{\rho V_F} w
\end{aligned} \quad (2)$$

Where V_F is the fractional volume open to flow, ρ is the fluid density, RDIF is a turbulent diffusion term, R_{SOR} is a mass source, A_z are the fractional area open to flow in x, y, z directions, (G_x, G_y, G_z) are body accelerations, (f_x, f_y, f_z) are viscous accelerations and (b_x, b_y, b_z) are flow losses in porous media. 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. The turbulent diffusion term in Equation 1. can be written as:

$$RDIF = \frac{\partial}{\partial x}(\upsilon_\rho A_x \frac{\partial p}{\partial x}) + R\frac{\partial}{\partial y}(\upsilon_\rho A_y R\frac{\partial p}{\partial y}) + \frac{\partial}{\partial z}(\upsilon_\rho A_z \frac{\partial p}{\partial z}) + \xi\frac{\rho\upsilon_\rho A_x}{x} \quad (3)$$

Where the coefficient $\upsilon_p = C_p\mu/\rho$, 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 [14].

3. Numerical Model Implementation

Using the Flow-3D software, Flow over the Doosti dam spillway was simulated. The suggested computational domain includes 150 meters before the spillway crest, the whole spillway structure and 250 meters after the spillway configuration. Furthermore, about 50 meters above the spillway crest have been considered in computational domain in vertical direction. Due to the comments in the Flow-3D user's manual [14], the Renormalization Group (RNG) turbulence model has been used for simulation which is the most accurate model available in Flow-3D software. The spillway geometry was created and defined using the AutoCAD 2007 software. Then the geometry was imported to the model as Stl file.

In the software, the flow domain is defined as a hexahedral in Cartesian coordinates; so, there are six different boundaries to be set. In this case flow data from free surface flows is desired; therefore, in vertical direction, the top boundary was set as atmospheric pressure, and bottom boundary was specified as wall. Since the purpose of these simulations is to model flow rate over a spillway with different water head levels, the upstream boundary was set as 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 downstream side. The boundary condition in n direction or the direction perpendicular to the flow, specified as wall at both sides. Figure 2. shows the boundary conditions set at each direction.

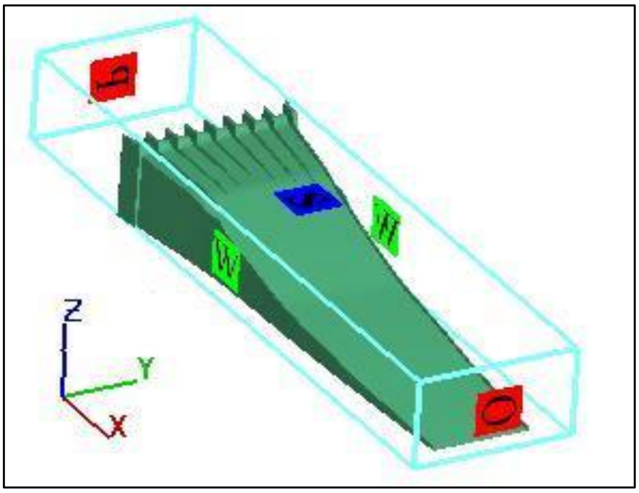

Figure 2. Boundary conditions set at each direction

In numerical modeling, it is very important to implement accurate initial conditions as closely as possible to the actual flow field. Rectangular regions were specified on the upstream and downstream of the spillway for initial condition, and the pressure considered as hydrostatic distribution in vertical direction. Rectangular fluid regions were specified on the downstream and upstream sides of the spillway as the same level as the specified fluid height at boundaries. Figure 3. shows the situation of the spillway at the beginning of the analysis

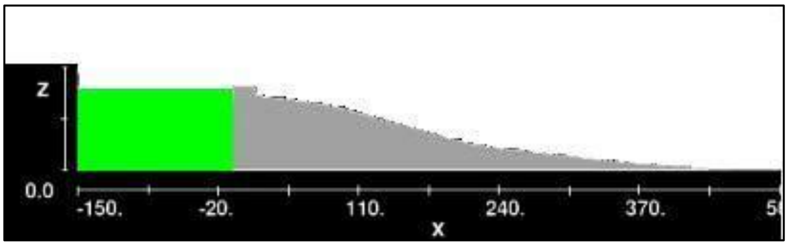

Figure 3. The situation of the spillway at the beginning of the analysis

Numerical modeling often 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 velocity and pressure. Determining the appropriate grid size is also an important part of any numerical simulation. Grid size can affect not only on the accuracy of results, but also on the simulation time. Therefore it is important to minimize the number of grids while including enough resolution to get significant features of geometry and the flow details sufficiently. In this study four different mesh types were considered. The grid size and the calculation time for each mesh type were presented in table 3. The results of each mesh compared with the experimental data for average flow depth in the discharge of 2660 m^3/s that is shown in Figure 4.

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

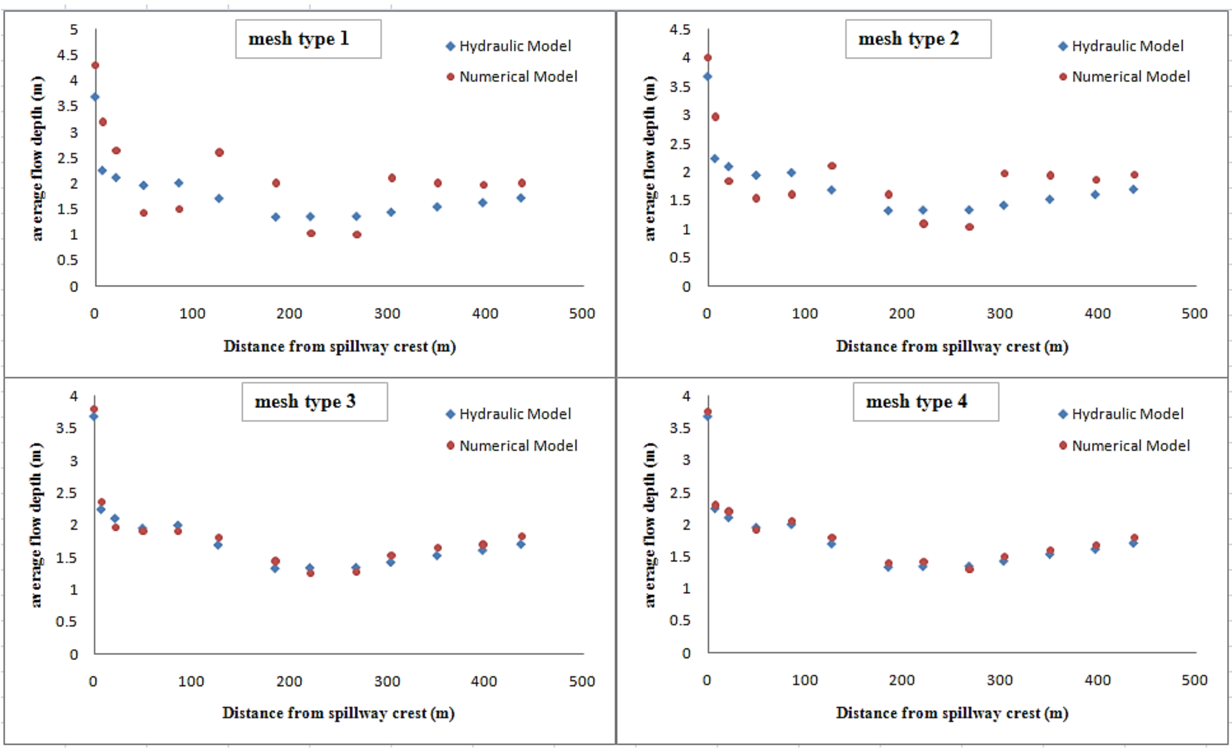

Figure 4. Mesh sensitivity analysis based on average flow depth

Table 3. Mesh type and the corresponding calculation time

Mesh type	Mesh size (x*y*z) (m)	Calculation time (min)
1	4*2.16*2	740
2	2*1.08*2	1011
3	2*0.72*1	1615
4	1*0.72*1	2240

4. Results

By three-dimensional numerical simulation, flow characteristics, including flow depth, velocity and pressure computed for five different flow rates. The numerical results were compared with the experimental data of the hydraulic model of Doosti dam spillway.

The average flow depth, the pressure and average velocity profiles along the spillway are shown in Figures 5 to 10. It can be observed that there is reasonably good agreement between the model results and hydraulic model measurements.

To quantify the accuracy and precision of the model, the Normalized Root Mean Square Error (NRMSE) and Coefficient of Determination (r^2) were calculated. The NRMSD represents the sample standard deviation of the differences between predicted values and observed values. These individual differences are called residuals when the calculations are performed over the data sample that was used for estimation, and are called prediction errors when computed out-of-sample. The NRMSD serves to aggregate the magnitudes of the errors in predictions for various times into a single measure of predictive power. NRMSD is a good measure of model accuracy. The coefficient of determination is a number that indicates the proportion of the variance in the dependent variable that is predictable from the independent variable. For a perfect fit between observed and predicted data, value of r^2 equals 1, and NRMSD should equal 0.0 respectively.

The calculated values for the NRMSE and r^2 have been presented in Table 4. The coefficient values show the reasonable precision of the numerical model.

Table 4. Evaluation of the model by some statistical coefficients

	Flow depth	Flow velocity	Flow pressure
NRMSE	0.0296	0.0437	0.0476
r^2	0.9196	0.8949	0.8354

After determining the flow characteristics, the cavitation damage can be predicted for the spillway. Occurrence of the cavitation damage on a spillway can be predicted using the cavitation index (Equation 4). To provide the structure safety, it is required to keep the flow cavitation index more than damage criteria everywhere on the structure [3].

$$\sigma = \frac{P - P_v}{\rho \frac{V^2}{2}} \tag{4}$$

Where V is reference flow velocity, P is reference flow pressure, P_v is water vapour pressure and ρ is water density.

The cavitation index equation (Equation 4) on a spillway surface can be expressed as:

$$\sigma = \frac{\frac{P_{At}}{\gamma} - \frac{P_v}{\gamma} + h\cos\theta}{\frac{V^2}{2g}} \tag{5}$$

Where P_{At}/γ is atmospheric pressure, which is considered 10.33 meter water height in the normal situation, P_v/γ is the water vapour pressure that is considered 2450 Pascal or 0.25 meter water height at 20 degrees of centigrade, θ is angle to horizontal axis.

Based on the flow cavitation index, five different levels have been considered for cavitation damage risk to a spillway structure (Table 5). The level intervals have been determined based on previous experiments and researches on the cavitation damage mechanism over dam spillways [15].

Table 5. Cavitation damage levels [15]

Level	Cavitation damage risk	Cavitation index
1	No cavitation damage	$\sigma > 1$
2	Possible cavitation damage	$0.45 < \sigma \leq 1$
3	Cavitation damage	$0.25 < \sigma \leq 0.45$
4	Serious damage	$0.17 < \sigma \leq 0.25$
5	Major damage	$\sigma \leq 0.17$

Due to numerical model, the values of water depth, velocity and piezometric pressure were calculated along the Doosti dam spillway for different flow rates. Then the cavitation index values were calculated according to corresponding flow velocity and piezometric pressure values. Figure 11. shows the cavitation index values for all flow rates.

According to the results, the spillway can be at the risk of cavitation damage, and as the flow rate increases the cavitation risk increases too. The serious damage may occur at ending parts of the spillway (distance of 300 m to 436 m from the spillway crest) for high flow rates respect to high flow velocities and pressure drops.

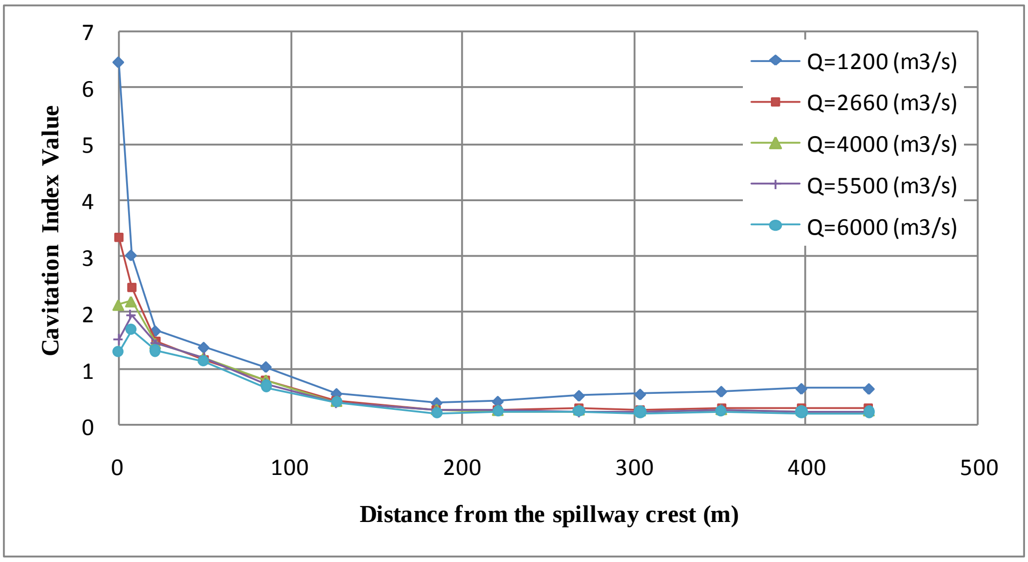

Figure 11. Cavitation index values along the spillway for different flow rates

5. Conclusion

In this paper, flow over the Doosti dam spillway located in Iran was simulated using the CFD method. The flow characteristics including flow depth, velocity and pressure computed for different flow rates. The numerical results agreed well with experiments, and Reasonable values for the normalized root mean square error and coefficient of determination indicated that the numerical model is accurate. Finally cavitation damage levels and locations for the spillway were predicted. Five damage levels from no damage to major damage have been presented based on cavitation index. Results showed that the spillway may be at the risk of cavitation damage, and the serious damage can occur at ending parts of the structure. By the results of this study, it is possible to find appropriate solutions in order to prevent damage with respect to the damage conditions in each part of the structure.

6. References

[1] Chow, V. T. “Open channel hydraulics”. McGraw- Hill Book Comp. New York, N Y, (2009).

[2] Kermani, E. Fadaei, G. A. Barani, and M. Ghaeini-Hessaroeyeh. "Prediction of cavitation damage on spillway using K-nearest neighbor modeling." Water Science and Technology 71, no. 3 (2015): 347-352.

[3] Falvey, H. T. Cavitation in chutes and Spillways. Engineering monograph No. 42, United States Department of the Interior – Bureau of Reclamation, Denver, Colorado, (1990).

[4] Hay, D. "Model prototype correlation: hydraulic structures." J. Hydraul. Eng. 113, no. 8 (1988): 899-907.

[5] Nie, Meng-Xi. "Cavitation prevention with roughened surface." Journal of Hydraulic Engineering 127, no. 10 (2001): 878-880.

[6] Dong, Zhi-yong, and Pei-lan Su. "Cavitation control by aeration and its compressible characteristics." Journal of Hydrodynamics, Ser. B 18, no. 4 (2006): 499-504.

[7] Frizell, K. Warren, Floriana M. Renna, and Jorge Matos. "Cavitation potential of flow on stepped spillways." Journal of Hydraulic Engineering 139, no. 6 (2012): 630-636.

[8] Savage, Bruce M., and Michael C. Johnson. "Flow over ogee spillway: Physical and numerical model case study." Journal of Hydraulic Engineering 127, no. 8 (2001): 640-649.

[9] Qian, ZhongDong, XiaoQing Hu, WenXin Huai, and António Amador. "Numerical simulation and analysis of water flow over stepped spillways." Science in China Series E: Technological Sciences 52, no. 7 (2009): 1958-1965.

[10] Zhenwei, M. U., Zhang Zhiyan, and Z. H. A. O. Tao. "Numerical simulation of 3-D flow field of spillway based on VOF method." Procedia Engineering 28 (2012): 808-812.

[11] Fadaei-Kermani, E., and G. A. Barani. "Numerical simulation of flow over spillway based on the CFD method." Scientia Iranica. Transaction A, Civil Engineering 21, no. 1 (2014): 91.

[12] 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, (1985): 1-19.

[13] Ho, H., Boyes, K., Donohoo, S. and Cooper, B. "Numerical flow analysis for spillways." Proc., 43rd ANCOLD Conf., Hobart, Tasmania, (2003): 24–29.

[14] Flow Science, Inc, "Flow-3D User's Manuals, Version 9.2", Santa Fe, NM, (2007): 110-114.

[15] Kermani, E. Fadaei, G. A. Barani, and M. Ghaeini-Hessaroeyeh. "Investigation of cavitation damage levels on spillways." World Applied Sciences Journal 21, no. 1 (2013): 73-78.

[16] Iran Water Research Institute (IWRI). "Doosti Reservoir Dam,"Hydraulic Department, Iran Ministry of Energy, Tehran, (2003).