Bulletin of Environment, Pharmacology and Life Sciences
Bull. Env. Pharmacol. Life Sci., Vol 3 (1) December 2013: 83-90

Online ISSN 2277-1808
Journal's URL:http://www.bepls.com
CODEN: BEPLAD
Global Impact Factor 0.533
Universal Impact Factor 0.9804

ORIGINAL ARTICLE

The Study of Cavitation Phenomenon in the Chute of Jarreh Dam by Applying FLOW-3D Model

Ebrahim Nohani[1*], Saman sakhaei [2], Ali Afrous[3]
1,2,3-Department of hydraulic Structures, Dezful Branch, Islamic Azad University, Dezful, Iran.
*Corresponding author Email: nohani.e@gmail.com

ABSTRACT
Usually Cavitation is the result of critical combination of velocity, pressure and vapor pressure. Bumps and unevenness on the concrete surfaces cause a deviation in the water flow lines and decrease the pressure in some areas. When water flows over a damspillway, the irregularities on the spillway surface will cause small areas of flow separation in a high velocity flow, and, in these regions, the pressure will be lowered. If the velocities are high enough the pressure may fall to below the local vapor pressure of the water and vapor bubbles will form. When these are carried downstream into high pressure region the bubble collapses giving rise to high pressures and possible cavitation damage. If the pressure drop is the result of velocity increase, threshold condition or threshold of cavitation (erosion) will be happened. Flow velocity, flow pressure, the resistance of the materials of the dam's spillway, and the rate of air flow over the spillway are of the most important parameters involved in the occurrence of the phenomenon. Generally, there are different methods for surveying the cavitation phenomenon, such as cavitation index. According to the present research and due to measurements resulted from Flow-3d Model and by considering the critical cavitation index (0.2), there is no possibility of cavitation occurrence for the chute of the Jarreh Dam. The result is very similar to the findings of the physical model constructed in the laboratory of the Water Research Institute.
Key words: Cavitation, FLOW-3D, Numerical Model, Chute.

Received 02/11/2013 Accepted 30/11/2013

INTRODUCTION

For several years, the events related to the problem of cavitation have attracted the attention of engenderers in different parts of the world. Cavitation happens in the spillways of high dams in the area where the flow velocity is high. Merely, what affects the phenomenon is not limited to velocity. The diversity of determinants requires a specific investigation regarding spillways' cavitation. In Iran, the phenomenon of cavitation of Karoon1 Dam causes some damages and problems (Figure1). In order to understand the mentioned phenomenon and the ways to prevent it, physical model are constructed in different research centers.

Figure1. Damage of cavitation-Karoon1 Dam

In high velocities, cavitation is a phenomenon that causes damages and cavities. Often, in a hydraulic system and due to a rise in the velocity, the pressure will be dropped to a level that is equal to the fluid vapor pressure in that condition. Such an event can also happen when flow lines deviate from their floors because of unevenness on the spillways. Actually, local pressure will be dropped to the level of fluid vapor pressure because of the deviation. In the case of occurrence of each of the two mentioned conditions, suddenly, the flowing fluid will be boiling and then it will be turned into vapor which makes some bubbles. After passing a short distance, the bubbles reach to high pressure areas and implode which generate noises. The imploding bubbles cause shock waves and strike to the border between the fluid and the structure. Consequently, erosion will be happened to the solid border (structure); in other words, since the bubbles' contact-surfaces with the spillways' floor are very small, a tremendous forced will be entered into the floor due to the implosions. Practically, after a while that the process continues, the spillways' floor will be eroded which causes large cavities [1].

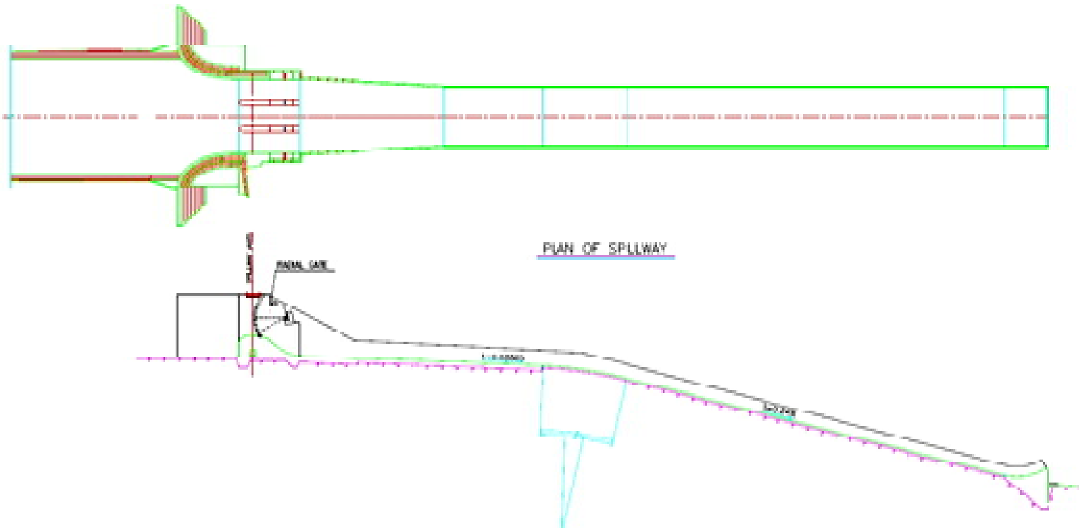
REVIEW OF LITERATURES

During the past years, researchers and scientists have attempted greatly to simulate free flows in open channels. Kazemi Torghaban & Barani [2] indicated that considering the simulations of flow over Dousti Dam's spillway and studies done around the world, Flow-3d Software has many advantages for modeling the flow over spillway due to the appliance of Volume of Fluid Model (VOF). Johnson & Swij [3] compared the characteristics of the flow over the free ogee spillway by applying once, the physical model and the other time, the numerical modeling. For numerical modeling, Flow-3d Software is used which solves RANS equations. In 2006, Johnson & Swij studied the characteristics of passing flow over ogee spillway in the presence of a downstream. In the mentioned modeling, Flow-3d software was used and findings showed that numerical modeling can foresight the flow behavior precisely [5]. Ferrari (2009) did a numerical simulation of flow with free surface over a sharp-crested spillway [6]. Etminan, et al (2011) considered the validity of numerical model by comparing the data with the laboratory findings of the Shahid Chamran University of Ahvaz and a high level of correspondence was resulted. It was indicated that the Flow-3d software can model non-falling stream over stepped spillway [7].

THE CHARACTERISTICS OF LABORATORY MODEL AND NUMERICAL MODEL

The Jarreh Dam's spillway model has been constructed in the scale of 1:50 in the hydraulic laboratory of Water Research Institute. The characteristics of the spillway model are as follows: A trapezoid-shaped gate with the length of 5.3m and 1.5m bed width is along the center of the spillway.

Ogee spillway is indicated through the following equation.($Y = 0.00531X^{1.85}$). A chute with the initial width of 64cm, initial length of 134cm, and the slope of %3.88 leads to the second part of the chute with the width of 50cm, length of 3.5m, and a constant slope of %24 by a sector with a radius of 400cm and 11/27°, and finally reaches to the cup-shaped thrower with a radius of 50cm and an angel of 45.5°. The horizontal length and the total height of the main flood drainage system from the initial part to the end of the thrower are orderly 649.38 and 128.8 cm. A view of the plan and longitude section of the Jarreh's spillway model is shown in the Figure2 [8].



.

Figure2. A view of the plan and longitude section of the Jarreh's spillway model

Erosion prediction

In order to predict erosion, we have to recognize the position of points in which there is a possibility of pressure drop to the level of water vapor pressure. In this regard, we can use the energy equation (Bernoulli equation) between two points in a constant flow. Actually this is a quantitative criterion. [9]

Equation 1: $$\rho g Z_0 + P_0 + \frac{1}{2}\rho V_0^2 = \rho g Z + P + \frac{1}{2}\rho V^2$$

The symbols are as follows:
ρ: Density of the fluid
g : acceleration of gravity
V & V_0 : Velocity at the radix point and the sightly point
Z & Z_0 :Balance at the radix point and the sightly point
Equation 1 can be written in the form of the dimensionless Equation 2.

Equation2: $$\frac{(P + \rho g Z) - (P_0 + \rho g Z_0)}{\frac{1}{2}\rho V_0^2} = 1 - (\frac{V}{V_0})^2$$

Often, we forebear the left sentences of the Equation 2 because of equality of Z & Z_0 balances and the negligible influence of the gravity. So we will have:

Equation3: $$C_p = \frac{P - P_0}{\frac{1}{2}\rho V_0^2}$$

It is called pressure parameter or Euler number. If P (fluid vapor pressure in the ambient temperature) is replaced by P_v we will have:

Equation4: $$\sigma = \frac{P_0 - P_v}{\frac{1}{2}\rho V_0^2}$$

This is called the erosion coefficient. So, σ is the ratio of necessary pressure drop for water evaporation to the potential of the flow pressure reduction by Kinetic energy.
Equation 4 can be written in the form of Equation 5 in the free surface flow over spillways by considering vertical curve.

Equation 5: $$\sigma = \frac{\frac{P_{Atm}}{\gamma} - \frac{P_v}{\gamma} + h\cos\theta \pm (\frac{h}{g} \times \frac{V_0^2}{r})}{\frac{V_0^2}{2g}}$$

γ: water weight
θ : chute's floor angel to the horizon
r : curvature radius of the vertical curve (symbol+ in convex curvature and symbol- in concave curvature)
$h\cos\theta$: depth of flow perpendicular to the floor

Erosion will occur when σ is equal to or smaller than σ_{cr} (critical erosion coefficient). The critical amount is often dependant on flow geometry, the shape and height of surface unevenness and the position of radix point wherein pressure and velocity are measured. According to the findings resulted from testing the model and real sample, different conditions are provided for determining σ_{cr} in diagrams and semi-empirical relationships.
If σ is smaller than the mentioned range, erosion will happen inevitably. To recognize the erosion potential in structures, designers calculate water surface profiles for different water flows. When the pressure on the floor and relative velocity are determined, the level of erosion for different points will be calculated and compared to the amounts of σ_{cr} in those points. In the case of having $\sigma \leq \sigma_{cr}$ as a result, there is a possibility of erosion in that area. Computation of erosion coefficient must be done for different water flow rates; because, the most critical status is not necessarily happens for the maximum water flow.

Surveying the erosion phenomenon along the chute of the Jarreh Dam's spillway

In order to survey and control the erosion, some information about the relative velocity and the pressure on the floor in different parts of the structure are required. Along the chute, 8 points are specified for obtaining the mentioned information. In order to calculate the erosion coefficient, Equation 6 is used.

Equation 6 :
$$\sigma = \frac{\frac{P_{Atm}}{\gamma} - \frac{P_v}{\gamma} + h\cos\theta}{\frac{V_0^2}{2g}}$$

In the Equation:

$\frac{P_{Atm}}{\gamma}$: The pressure of the environment which is equal to 1 atmosphere or 10.33m of water column in laboratory conditions.

$\frac{P_v}{\gamma}$: The amount of fluid vapor pressure which is equal to 0.33m of water height in the relative temperature of 25 C°. In order to have a valid calculation, the mentioned amount is considered as 1m of the water height.

$h\cos\theta$ or $\frac{P_0}{\gamma}$:The pressure amount on the structure (corresponding pressure of the water height) which is calculated in different points.

$\frac{V_0^2}{2g}$: Height corresponding to the velocity (by meter) in the sightly point.

By entering the numerical amounts into the Equation 6, Equation 7 will be resulted.

Equation 7:
$$\sigma = \frac{(10.33 - 1 + \frac{P_0}{\gamma})}{\frac{V_0^2}{2g}} = \frac{2g}{V_0^2}(9.33 + \frac{P_0}{\gamma})$$

Ramhormoz City is located 190 meters above the sea level ,but the physical model constructed in Tehran has considered the number as 1190 meters. So, this 1000-meter-difference will change the Equation 7 as follows:

Equation 8:
$$\sigma = \frac{(10.33 - 1 + \frac{P_0}{\gamma})}{\frac{V_0^2}{2g}} = \frac{2g}{V_0^2}(9.71 + \frac{P_0}{\gamma})$$

Flow-3d Model can be applied to a lot of physical patterns such as shallow waters, viscosity, cavitation, turbulence, porous zone, etc. The model has function for casting, process engineering, hydraulic, environmental, aerospace, marine science, oil, gas, etc. Flow-3d Model has a high capability of simulating water flow, and sediment transport and scour. Generally it is a suitable model for hydraulic simulation. What differentiates it from other similar models is using VOF method for the prediction of fluid level and its combination with FAVOR (Fractional Area-Volume Obstacle Representation) method for determining rigid boundaries. Simple cubic network is one of the other characteristic of Flow-3d model [8].
At the first glance, the cubic network may be considered as a limitation. But the easiness of the network production, proper order, occupying less memory, and the application of two useful tools of VOF and FAVOR in the Flow-3d model are the advantages of the network.
Recently, Flow-3d model has been used in many fields because of its capabilities in hydraulic simulations and producing acceptable responds. Regarding hydraulic analysis, one of the advantages of the software is using VOF in the modeling of free surface flows which has remove the obstacles we were faced with in the previous methods (based on trial and error)[8].
In the present paper, 6 water flow rates (500, 1000, 1500, 1900, 2100, 2850 cubic meters per second) related to the Jarreh's spillway are used for simulation. Stl file format is used as input for modeling the Jarreh Dam's spillway. The process is as follows:

Within the CAD file, the plan and longitudinal profile of the spillway are drawn by using 3D environment of AutoCAD. The created file will be saved as a *.stl file and called in the software. Figure 3 shows a view of the called spillway in the model.

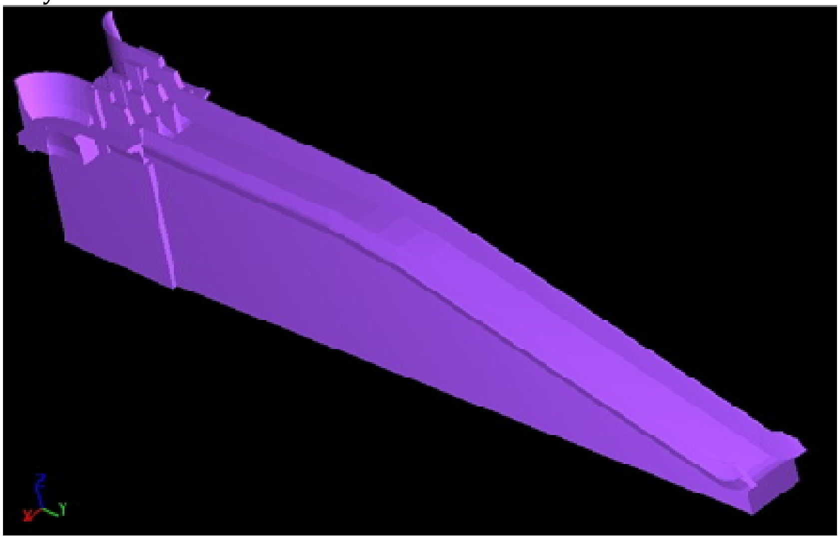


Figure 3 .view of spillway in the model

During the process, necessary data such as shape, simulation time, type of the fluid, turbulence calculation equations, etc. will be obtained and finally they will be compared with physical data of chute. The present paper has applied RNG model for turbulence calculation.

One of the important factors regarding simulation is determining the framework of the computational grid. The number of cells in a mesh is dependent on the area and determined size. The numbers will show the level of accuracy, the end time, and the result. Since the type and density of meshing will affect the accuracy and determined time, a suitable one is required for a valid calculation and the fulfillment of the predicted end time. Figure 4 shows a view of spillway meshing in the software.

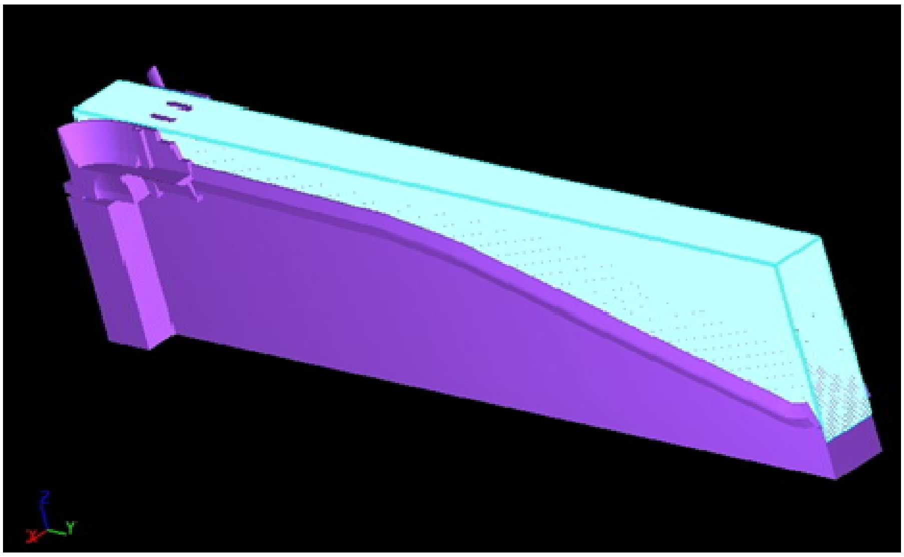


Figure 4- A view of spillway meshing in the software

In order to determine the Boundaries conditions, volume flow rate is used for the inflow point of the spillway (Ymin). Wall border condition (Wall) is used for bulwarks (Xmin, Xmax) and channel floor (Zmin), because fluid velocity over solid border is equal to zero. In the upper border of the spillway (Zmax), symmetry condition (Symmetry) is used and in the outflow point (Ymax), outflow condition is applied (Figure 5).

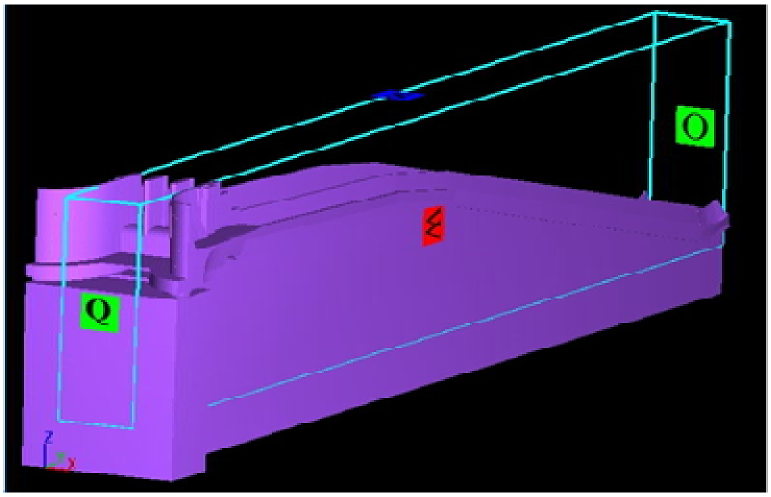


Figure 5. Boundaries conditions applying in the software

After inserting necessary data into the software and doing simulation, the validity of models calibration based on roughness will be tested and compared with the results of the physical model.

A comparison between numerical and laboratory models

Considering the pressure amounts and velocities obtained from the software of calculating water flow rates (500-2850 cubic meters per second) in the lateral point and applying them to the Equation 8, the amount of erosion coefficient in 8 points of the chute of the spillway has been calculated. The results shown in the Table 1 and the curve of erosion coefficient changes along the chute for 6 water flows in Figure 6 are compared with the data obtained from the physical model.

Table1- calculating erosion coefficient along the chute for 6 water flow rates (500-2850 m^3)

point	z	y	Q= 500 m3		Q= 1000 m3		Q= 1500 m3		Q= 1900 m3		Q= 2100 m3		Q= 2850 m3	
			physical model	Flow3d	physical model	Flow3d	physical model	Flow3d	physical model	Flow3d	physical model	Flow3d	physical model	Flow3d
G	481	25.44	0.94	0.91	0.89	0.94	0.96	1.08	1.06	1.16	1.11	1.25	1.2	1.35
Percentage of difference			3.30		5.32		11.11		8.62		12.61		12.50	
H	479	76.98	0.94	0.86	0.72	0.81	0.73	0.81	0.74	0.79	0.72	0.80	0.75	0.8
Percentage of differencd			9.30		11.11		9.88		6.33		11.11		6.67	
I	477	128	0.86	0.78	0.67	0.7	0.66	0.65	0.69	0.70	0.68	0.69	0.66	0.67
Percentage of difference			10.26		4.29		1.54		1.43		1.47		1.52	
J	475	154.17	0.82	0.74	0.63	0.65	0.55	0.61	0.55	0.59	0.52	0.59	0.55	0.61
Percentage of difference			10.81		3.08		9.84		6.78		13.46		10.91	
K	470	178.99	0.61	0.56	0.5	0.51	0.47	0.5	0.5	0.50	0.46	0.49	0.46	0.48
Percentage of difference			8.93		1.96		6		0.00		6.52		4.35	
L	460	220.66	0.41	0.41	0.32	0.35	0.31	0.35	0.32	0.34	0.3	0.33	0.32	0.35
Percentage of difference			0.00		8.57		11.43		5.88		10.00		9.37	
M	445	283.16	0.3	0.31	0.25	0.27	0.24	0.25	0.24	0.26	0.23	0.26	0.24	0.26
Percentage of difference			3.23		7.41		4		7.69		13.04		8.33	
N	431	341.49	0.29	0.29	0.28	0.27	0.22	0.23	0.22	0.22	0.2	0.22	0.2	0.22
Percentage of difference			0.00		3.7		4.35		0.00		10.00		10.00	

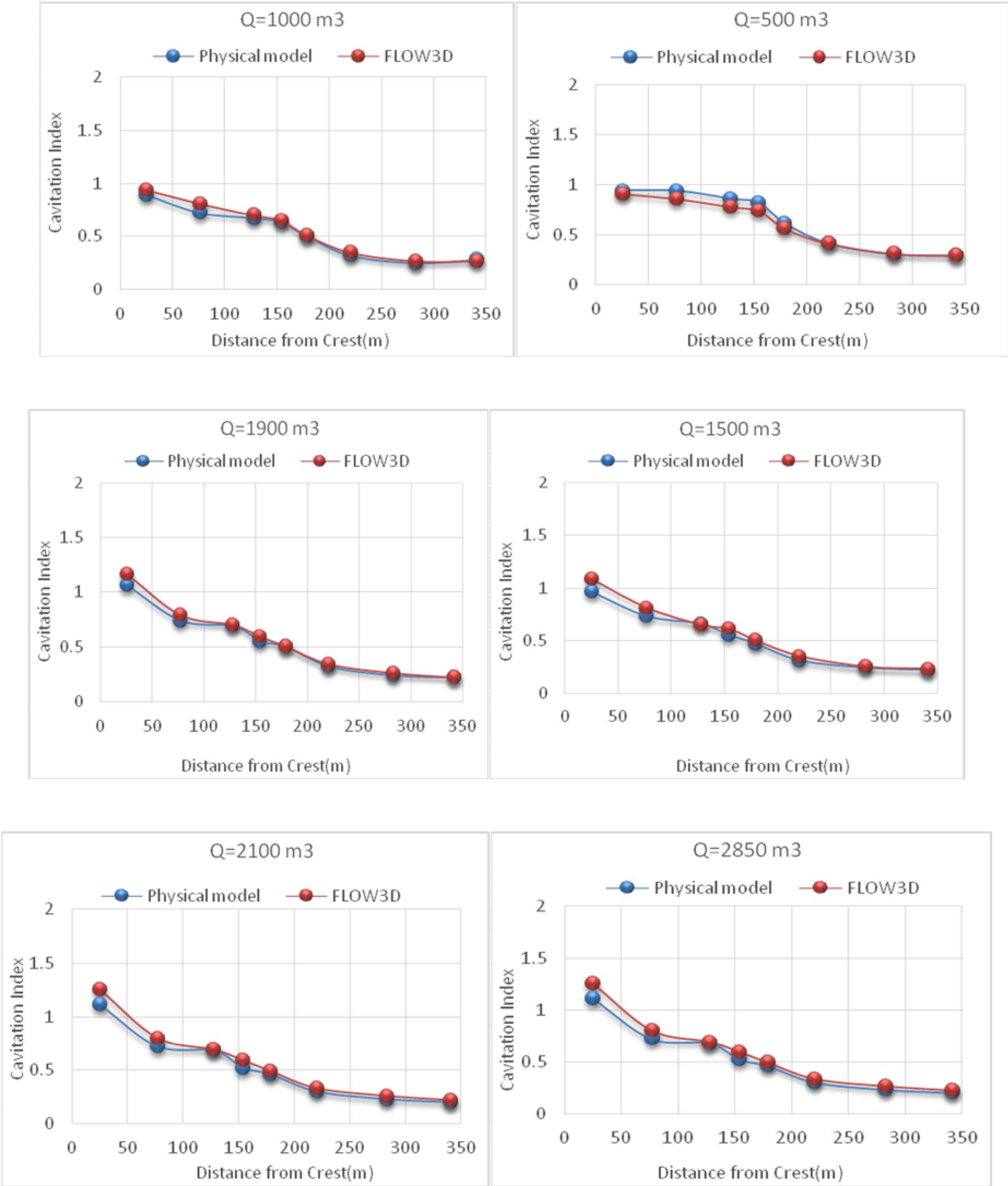


Figure 6. curves of erosion coefficient changes along the chute for 6 water flows

CONCLUSION

1- Erosion coefficient of the flow is calculated by the help of parameters of velocity and static pressure in the lateral point for 6 water flow rates. The findings indicate that unlike the velocity, erosion coefficient changes are declining along flow over the chute. The minimum calculated erosion coefficient is equal to 0.22, in the water flows of 1900, 2100, and 2850 cubic meter per second and at the ending part of the chute. According to the calculated erosion coefficient from the numerical model and critical erosion coefficient, there is no possibility of erosion phenomenon along the chute. (δ>0.2)

2- It seems that using blower structures for preventing damages and possible consequences of erosion is not necessary, but definitive recommendation regarding using such structures is dependent on other factors like the air concentration in the vicinity of the concrete surface of the chute, the quality of the concrete surfaces, and the quality of the contraction joints which are of great importance.

3- The results show that the most critical state of erosion can happen for any water flow rate which is not necessarily the maximum one of the floodwater. So, in surveying the cavitation of spillways and chutes we should not merely focus on maximum water flow rate, because erosion can happen even for lower water flow rates.

4- By a comparison between data of laboratory model and Flow-3d model, it can be concluded that the software provides acceptable results in hydraulic calculations.

REFERENCES

1. Erfanian Azmoodeh, M.H, 2011. Simulation of cavitation over spillway and chute of Upper Gotvand Dam by applying Flow-3d software, MA Thesis, Shahid Chamran University of AHvaz.
2. Dehdar Behbahani, Sadeq, 2011. A hydraulic survey of Balaroud Dam's spillway flow by applying Flow-3d model and comparing it with the physical model, MA Thesis, Shahid Chamran University of Ahvaz.
3. Kazemi Torqaban, Mostafa & Brani, Qolamabbas, 2012. Determining the hydraulic conductivity coefficient of Dousti Dam by applying Flow-3d software, The 9th Congress of Civil Engineers, Isfahan University of Technology.
4. Michael C. Johnson, M.ASCE; and Bruce M. Savage, M.ASCE, 2006. Physical and Numerical Comparison of Flow over Ogee Spillway in the Presence of Tailwater, Journal of Hydraulic Engineering ,vol 132,pp.1353-1357.
5. Ferrari, A., 2009. SPH Simulation of Free Surface Flow Over a Sharp Crested Weir,"Department of Civil and Environmental Engineering, University of Trento, Via Mesiano 77, I 38100 Trento, Italy.
6. Etminan, Hamed & Mousavi Jahromi, Habib, , 2012. "Surveying the effects of deflector on the length of blowing point in stepped spillway by applying the Flow-3d software", The 9th Congress of Civil Engineers, Isfahan University of Technology.
7. The final report of hydraulic model of flood drainage system of Jarreh reservoir dam, Water Research Institute, Ministry of Energy.
8. FLOW-3D user manual, ver 9.3.
9. QasemZadeh, Firouz, 2013. Simulation of hydraulic issues and structures with Flow-3d software", Tehran Nouavar Publication.

Citation of this article
Ebrahim Nohani, Saman sakhaei, Ali Afrous .The Study of Cavitation Phenomenon in the Chute of Jarreh Dam by Applying FLOW-3D Model. Bull. Env. Pharmacol. Life Sci., Vol 3 (1) December 2013: 83-90