Numerical study of the passing flow over spillway with emphasis on the potential of cavitation occurrence

Hamzeh Ebrahimnezhadian[1], MirAli Mohammadi[2], Ramin Abdullah Kookhi[3] Ako Seyedi[4]

1- Ph.D. Candidate in Water & Hydraulic Structures, Dept. of Civil Eng., Faculty of Eng., Urmia University, Urmia, Iran

2- Associate Prof. in Civil Eng., Water & Hydraulic Structures, Dept. of Civil Eng., Faculty of Eng., Urmia University, Urmia, Iran

3- Ph. D. Candidate in Water & Hydraulic Structures, Dept. of Civil Eng., Faculty of Eng., Islamic Azad University, Arak, Iran

4- MSc in Civil Engineering-Water, Islamic Azad University, Maragheh, Iran

Email: H.ebrahimnezhadian@gmail.com

Abstract

Water has high velocity passing through spillways or bottom channels of dams. Under this condition, it would have a great deal of kinetic energy; therefore, energy dissipater structures must be used in order to dissipate such destructive energies. Analyzing the flow passing through spillways is typically done by analytical relations and physical models. However, nowadays thanks to advances in computer sciences, using numerical models has grown noticeably, because of using less time and cost compared to physical models. In this study, flow characteristics, including pressure distribution and average flow velocity, were studied in different points of flood evacuation system, and were compared with the results of physical model. Then, using the results of pressure and velocity, potential occurrence of destructive cavitation was studied on chute of spillway. The results indicated acceptable compatibility between the results of numerical model and those of experimental model; therefore, it can be argued that this software has an acceptable ability in simulating free surface flows, particularly in velocity and pressure field. RNG K-ε turbulence model, among other turbulence models, has the highest compatibility in simulating flow characteristics. The study of results showed that at a certain span in the length of spillway, cavitation index is set lower than the critical point, and the occurrence of cavitation is strengthened from this point onwards and the risk of cavitation occurrence increases. This necessitates the use of aeration equipment in such areas in order to prevent the occurrence of cavitation.

Keywords: Ogee spillway, velocity and pressure distribution, cavitation indicator, Flow-3D software.

1. INTRODUCTION

Spillways are one the most important components of dams, and at times when dam's reservoir cannot hold flood water due to its limited capacity, spillways move the flood water to downstream in a safe way. The reason of failure in most of the dams around the world is because of inadequate designing of spillways. Spillway geometry depends on many factors including topography, shape of valley, bedrock, type of dam, and etc. The process of flow passing over spillways typically causes remarkable destructive energy at the downstream of spillways by transformation of upstream sub-critical flow to downstream supercritical flow. One of the issues that happens in high spillways is cavitation. There are typically uneven surfaces on spillways that when water flows on them and with the increase in flow velocity (more than 10 m^3/s), the flow is separated from spillway, which causes localized pressure reduction and results in cavitation and erosion in concrete surface, and finally ends in destruction of the structure. This destructive phenomenon has caused heavy losses in many dams in different parts of the world. In Iran, several times it destroyed Shahid Abbaspoor Dam's spillway, whose reconstruction and repair cost was enormous. There are several ways to control cavitation. Cavitation control by changing the geometry of spillway and using concrete spillway surface is a primary corrective action. The most effective way in controlling cavitation is artificial aeration in the flow. This is accomplished by embedding aeration in spillway. Hence, checking the passing flow over the spillway has always been one of the most important steps in designing dams, but unfortunately, in most dam projects, investigating the passing flow over the spillway is not achieved, due to high costs and time-consuming physical models. Replacing hydraulic models with numerical models has always been one of the points which has progressed substantially in analyzing hydraulic issues, thanks to the advances of computer science and development of computational fluid dynamics (CFD). Using physical models, despite a way of analyzing the passing flow over the spillway, is costly and time-consuming. However, using mathematical models that simulate hydraulic phenomena could help make more converged results, without consuming lots of time or money. Thus, one of the best tools for the analysis of the passing flow over the spillways is using mathematical models. Obviously, the use of numerical models has its own specific problems including calibration, having powerful hardware for simulation, and choosing the appropriate methods for better convergence of the results. Nevertheless, due to savings in costs and time, using the appropriate numerical model can always help. [1]

Using numerical model in simulating the passing flow over spillways was done two dimensionally for the first time in 1965 by Cassidy to determine the pressure on the spillway crest based on the potential flow theory. The results of this study indicate that first, there is a good match between experimental and numerical model data. Second, the viscosity does not affect the determination of free surface. Hu et al. (2003), examined pressure changes on spillway crest, and hydraulic conductivity of ogee spillway in the upstream head using Flow-3D two-dimensionally and three-dimensionally, and compared the results with the results obtained from the USACE. The results corresponded with USACE data. However, in both two-dimensional and three-dimensional models, numerical model predicts slightly more negative pressure on the spillway crest. In Iran, Kavianpoor et al. examined the process of static pressure changes along stepped spillway of Siah Bishe Dam using Fluent software. Experimental model was used to verify numerical model results, and the results indicated acceptable convergence. [2]

2. Geometry and Hydraulic Characteristics of Flow Field

Azad dam is a rockfill dam with clay core. The height of the dam from its foundation is 125 meters and the length of the dam crest is 600 meters. It has a body capacity of 85,000,000 m3 and reservoir capacity of 300,000,000 m3. The dam's catchment area is 1007 km2. Catchment gradient is 20%, the average gradient of river is 81%, and the main river length is 69 km. The purpose of this dam is to transfer 250,000,000 m3 per year to the plains of Qorveh and Dehgolan in East Kurdistan. A further objective of this dam is to release 50,000,000 m3 of water per year for environmental water rights and agricultural riffle. Azad River is one of the main branches of Sirvan River. The average annual input of this river at the dam axis is 376,000,000 m3. Hydraulic model of flood discharge in Azad dam with a scale of 1 to 33 was built and installed by the Water Research Institute. Azad dam's flood evacuation system includes: 1- input channel 2- valved ogee spillway 3- chute 4- flip 5- riffle flip, which are built on the left side of the dam. Three discharges of 1226, 1545, and 2990 m3/s (flood discharge with 1000-year and 10000-year return and probable maximum flood respectively) were measured.

3. Numerical Solution of the Flow Field

3.1. Prevailing equations in flow filed by Flow-3D numerical model

Flow-3D is a powerful software specified for computational fluid dynamics which has been designed by Flow Science Company. Solving equations in this application is based on the finite volume method. One of the major features of this software for hydraulic analysis, is the ability to model the flows with free surface. The interface between gas and liquid, is the flow with free surface. In this software, the free surface is modeled by VOF technique that was developed in 1981 by Hirt and Nichols. Also in this software, all equations are formulated with space and volume of penetration rate functions. This formulization is called FAVOR technique which was presented in 1985 by Hirt and Lysylyan.

The prevailing equations in fluid flow include the continuity equation and the momentum equation for incompressible turbulent flow with constant density and viscosity as expressed in equations (1) and (2) [4].

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

Where V_F is fractional volume of fluid, ρ is fluid density, RDIF is permeability turbulence, RSOR is mass source, u, v, and w are velocity components in Cartesian coordinates (x, y, z) or cylindrical coordinates (z, θ, r) respectively. A_x, A_y, and A_z are respectively the space of element in vertical direction to x,y, and z. ξ and R coeficient are related to system coordinates and their value in Cartesian coordinates are:R=1 and $\xi = 0$

$$\frac{\partial u}{\partial t} + \frac{1}{V_F}\left[u A_x \frac{\partial u}{\partial x} + v A_y R \frac{\partial u}{\partial y} + w A_z \frac{\partial u}{\partial z} \right] - \xi \frac{A_y v^2}{x V_F} = -\frac{1}{\rho}\frac{\partial P}{\partial x} + G_x + f_x - \frac{RSOR}{\rho V_F} u$$

$$\frac{\partial v}{\partial t}+\frac{1}{V_F}\left[uA_x\frac{\partial v}{\partial x}+vA_yR\frac{\partial v}{\partial y}+wA_z\frac{\partial v}{\partial z}\right]-\xi\frac{A_yvu}{xV_F}=-\frac{1}{\rho}\left(R\frac{\partial P}{\partial y}\right)+G_y+f_y-\frac{RSOR}{\rho V_F}v$$
$$\frac{\partial w}{\partial t}+\frac{1}{V_F}\left[uA_x\frac{\partial w}{\partial x}+vA_yR\frac{\partial w}{\partial y}+wA_z\frac{\partial w}{\partial z}\right]=-\frac{1}{\rho}\frac{\partial P}{\partial z}+G_z+f_z-\frac{RSOR}{\rho V_F}w \qquad (2)$$

In these equations G_x, G_y, and G_z are mass acceleration in the direction of x, y, and z, and f_x, f_y, and f_z are accelerations of gravity in the direction of x, y, and z, and the last term in the right side is related to the injection of mass at zero velocity.

3.2. DEFINITION OF SOLID BORDERS GEOMETRY

The present numerical study simulated Azad dam's spillway (which is a type of ogee spillway with flip bucket) using Flow-3D. To create a geometric model and define the geometry of solid boundaries, three-dimensional Auto CAD software was used. For a suitable convergence of the solution field, 1500000 computational volumes of rectangular adaptive type were used. Because of the turbulence of the discharge passing over the spillway, three-dimensional flow field has been solved by RNG turbulence model. Apparently, the right number of computational volumes of the issue, was viewed and chosen after performing various models in the software. Furthermore, in 'Boundaries', the boundary conditions of the issue were determined, and volume flow rate was chosen as the input border. In this section, input flow discharge was set and defined for the software, and outflow was chosen as the output border. Wall was chosen as the bottom border and the side borders. Wall boundary conditions or wall are used to separate liquids from solid border. In this boundary condition, the vertical and tangent velocity on the boundary wall is considered zero. Symmetry was chosen for the upper part that is in connection with the atmosphere, and in fact the flow has free surface. Figure (1) shows the flow geometric model and boundary conditions. [5]

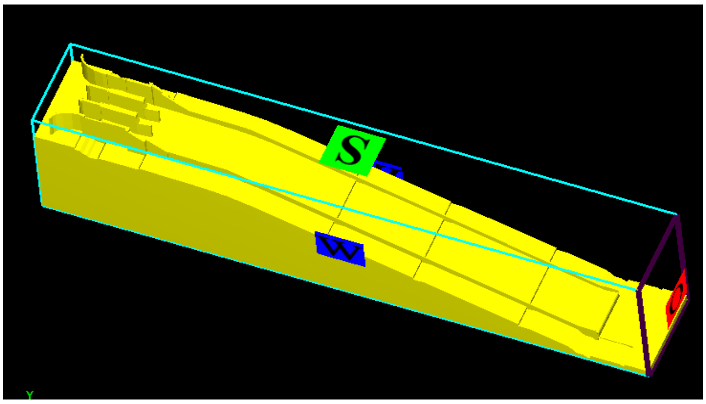


Figure 1 - View of the three-dimensional geometry and boundary conditions used by the numerical model of Azad dam's spillway

Table 1 – Specifications of numerical model of Azad dam's spillway

Meshing	Type of Model	VOF
	Type of Mesh	Adaptive Rectangular
	Number of computational volume	1500000
Boundary condition	Body of spillway	Solid
	Side boundaries	Wall
	Input	Volume Flow Rate
	Output	Outlet
Equation	Turbulence	$RNG\ K-\varepsilon$
	The Algorithm of Solving Pressure Equations	GMRES
	Free surface Model	Fluid Volume Pattern

4. Analysis of the Results

4.1. Changing the Froude number

One of the most important parameters in studying the pattern of passing flow over spillways is Froude number. Froude number is important because the process of passing flow over spillways is accompanied by transformation of subcritical flow at upstream to supercritical flow at downstream. In this section, using Flow-3D numerical model, the Froude number of passing flow over the spillway of Azad dam was studied.

In figure (3) it can be seen that the regime of passing flow over ogee spillway, has a suitable pattern in accordance with the process of passing flow over the spillways. According to the values of the Froude number, initially the flow regime was subcritical, and with increasing distance from the flow and chute, the flow regime goes toward supercritical. It is obvious that with the flow passing over the spillway and increase in discharge, velocity is always increased. Similarly, the changes in the Froude number significantly increase, like the speed along the flow on chute. The difference between the changes in Froude number and the changes in velocity is that as the discharge increases, flow velocity is also increased. However, as it can be seen in the graph, the increase in discharge, does not necessarily result in the increase in Froude number.

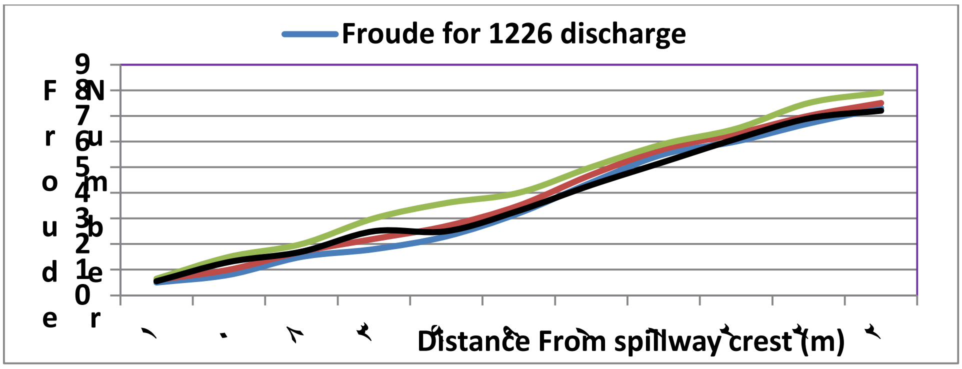
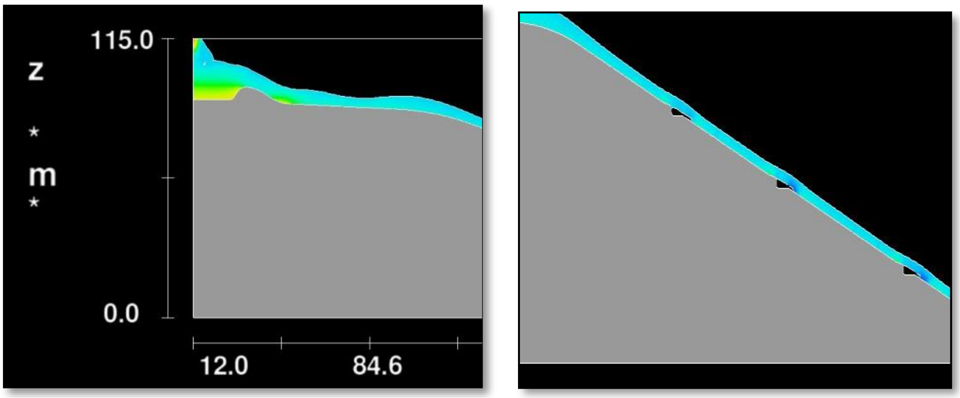


Figure 3- The comparison of Froude number changes over the spillway of Azad dam for critical discharges

4.2. The manner of changes in static pressure and average velocity

The results of this study include the average static pressures and flow velocity in mentioned sections along the spillway of Azad dam. For three discharges of 1226, 1545, 2290 m3/s (flood discharge with 1000-year and 10000-year return and probable maximum flood discharge respectively) static pressure and flow velocity were measured. Similarly, the static pressure and flow velocity in the numerical model at specific points of the mentioned discharges were measured by the software.

As shown in Figure 4, the pattern of passing flow over the spillway for the largest discharge has a good match with the natural mode, so that in upper depths of the passing flow, the static pressure increases. Due to the increasing depth of flow and pressure distribution in the areas before the ogee spillway, the pressure in this area is higher compared to other areas, while on the chute, what happens is the reduction of flow depth, followed by reduction of the pressure. This reduction is normal up to this point and happens in passing of the flow over all ogee spillways, and the reason is that, by increase in velocity, the flow depth is reduced and the pressure decreases. However, the most critical pressure happens when flow pressure has a tendency toward negative pressure. This mode that is commonly called suction, increases the likelihood of cavitation occurrence in the spillway structure. Figure 4 (b) shows the reduction of pressure on the chute and the possibility of formation of negative pressure in this range. [6]



a) Pressure changes near aeration chute b) Pressure changes on ogee spillway crest Figure 4- The manner of changes in the static pressure on the spillway and chute of Azad dam (for maximum discharge)

According to the pattern of the passing flow over Azad dam's spillway, the changes of flow velocity are in way that at the beginning of input channel, because of the depth of the water in the input channel, the flow velocity is much lower than the other parts of the spillway. The reason for this drop in velocity is that the input flow of the spillway encounters the lump of ogee spillway. The flow at the beginning of flood evacuation system in input channel is quiet and dominant to the spillway. As it can be seen from the changes of Froude number, the flow is almost subcritical, and with passing over the spillway and with gradual increase in velocity, it changes to supercritical. Figure (5).

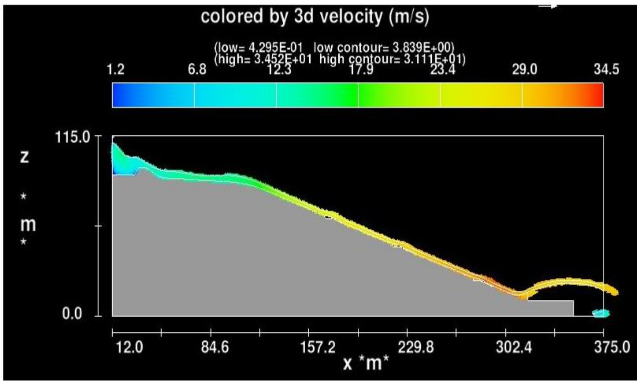


Figure 5- The changes of average velocity over the spillway for discharge of 2290 (maximum probable flood) by Flow-3D model

4.2. EVALUATION OF THE RESULTS OF THE NUMERICAL MODEL

To verify the results of the numerical model, the results of experimental model were used. The choice of three discharges was to verify the effects of the results of maximum, minimum and intermediate discharges. Therefore, the numerical static pressure, and experimental static pressure, and experimental and numerical velocity were calculated, and the graphs of velocity and pressure changes for three above mentioned discharges were drawn.

4.2.1. THE PRESSURE OVER SPILLWAY

According to evaluation of the charts, it can be said that there is a good match between experimental data and numerical data. At the beginning, the static pressure is high, and near the chute, the pressure is reduced, and finally, reaches its peak in flip bucket. It must be pointed out that, for larger discharges, the pressure increased. In aerator positions, the pressure increased and then decreased.

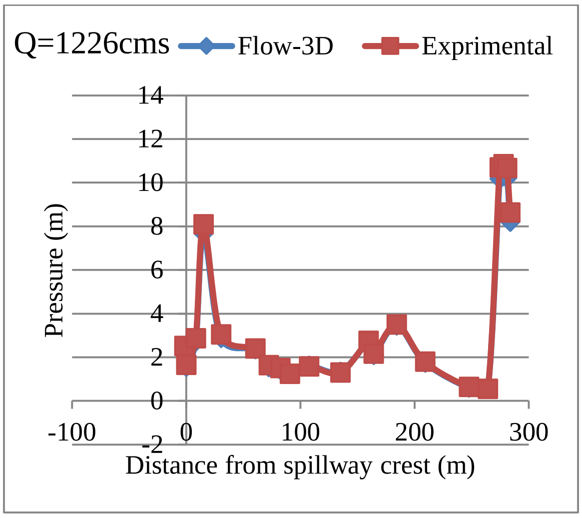


Figure 4-a) The comparison between numerical and experimental static pressure for discharge (1226 m^3/s)

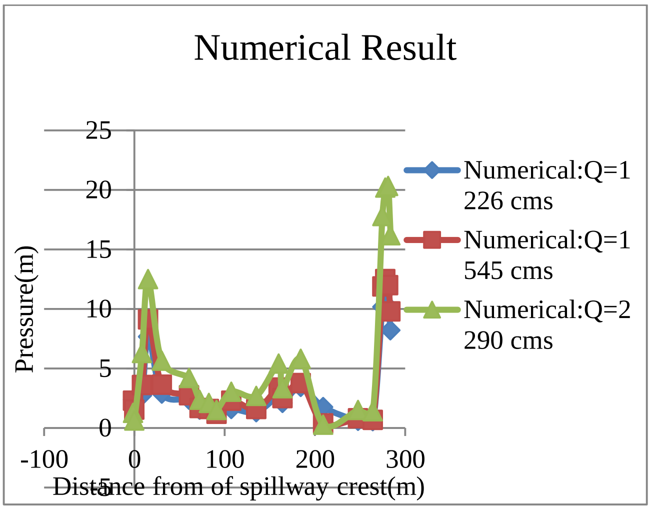


Figure 4-b) The comparison between numerical static pressure for three discharges (1226, 1545, 2290 m3/s)

4.2.2. FLOW VELOCITY OVER THE SPILLWAY

The evaluation of the results of velocity over the spillway indicates that with increasing distance from the spillway crest, flow velocity increased and reached its peak in flip bucket. The study of the results showed that with increasing discharge, flow velocity also increased.

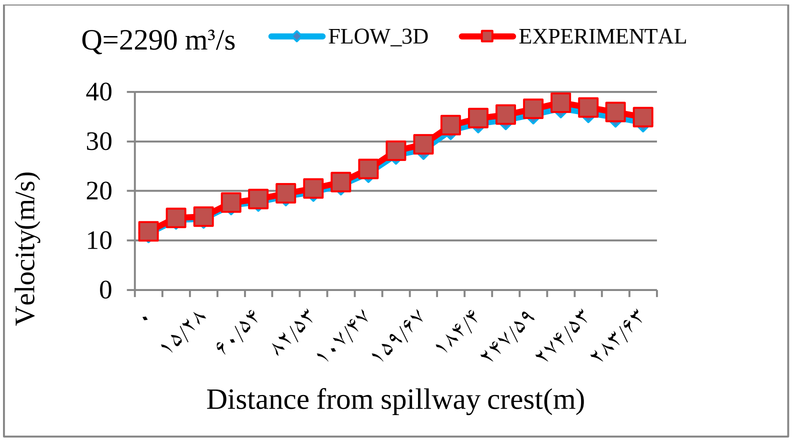


Figure 5- The comparison between numerical and experimental velocity for (2290 m^3/s)

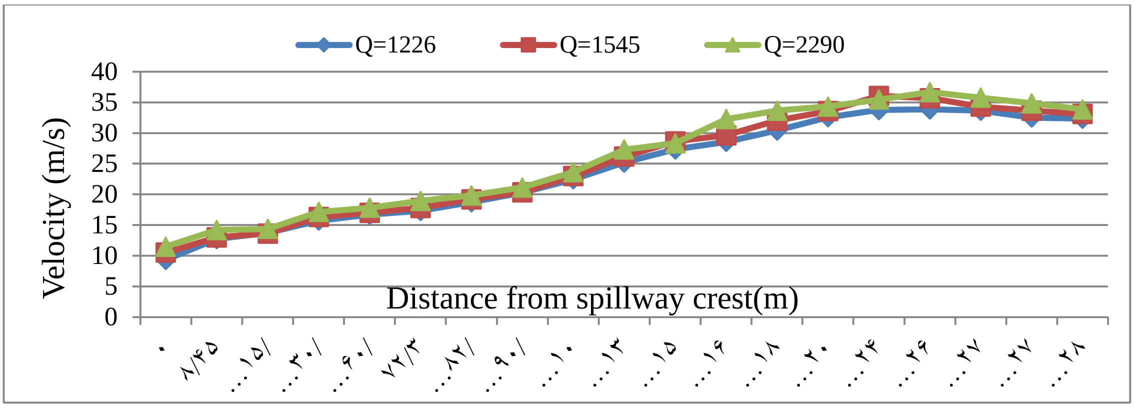


Figure 6- The comparison between numerical velocity for three discharges (1226, 1545, 2290 m³/s)

4.2.3. THE STUDY OF CAVITATION CHARACTERISTIC OVER THE SPILLWAY

4.2.3.1. CAVITATION INDEX

The main parameter for describing cavitation, is cavitation index, which is in fact a special form of Euler number.

$$\delta = \frac{(P_0 - P_V)}{\frac{1}{2}\rho u_0^2} \qquad (3)$$

Where P_0 is a characteristic pressure of the flow, u is a characteristic velocity of the flow, ρ is the density, and P_V is the vapor pressure of the fluid.

Using velocity and static pressure measurement results, in 12 cross sections along the chute for three discharges, (from 1226 to 1800 m3/s), cavitation coefficient was calculated. The results and change curves of cavitation coefficient are indicated in Figure (7). Using the results, test parameters (static pressure and velocity) of the chute cavitation coefficient along the chute were calculated. From the beginning of the chute, to a distance of 160 meters from the threshold of the spillway, cavitation coefficient ($\delta \geq 0.25$) is above critical line.
At a distance of about 115 meters from the chute ramp leading to the flip bucket, corrosion coefficient ($\delta \leq 0.25$) is below the critical line. In order to calculate corrosion coefficient (cavitation index) using numerical models over the chute of Azad dam's spillway, for three critical discharges, velocity and average pressure in 12 cross sections were calculated.

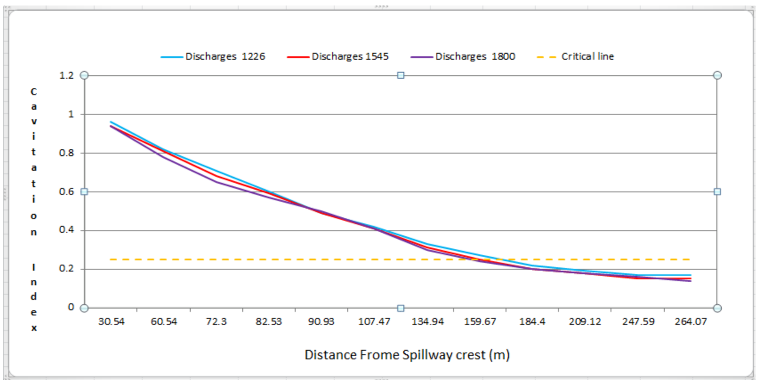


Figure 7- The comparison of curves of cavitation coefficient of Flow-3D model along the chute for three critical discharges

The important point is that in studying the cavitation coefficient over the chute for the discharge of maximum probable flood (2290 m3/s), because of the conditions of the passing flow, in experimental model, 1800 m3/s was used.

As it can be seen, cavitation coefficient acts contrary to the manner of velocity and pressure changes. The notable point is that with distancing from the spillway over the chute, cavitation coefficient has decreased. Like the changes in cavitation coefficient in physical model, in numerical models likewise, at a distance of approximately 160 meters from the threshold of the spillway, the corrosion coefficient is below the critical line, thus the occurrence of cavitation from this point onwards gains strength and the risk of cavitation occurrence increases. Therefore, in order to remove the corrosion on the surface of the chute, two aeration systems were designed and performed, one at a distance of 165 meters from the threshold of the spillways, and the other at a distance of 210 meters from the spillway.

5. Conclusions

In general, the results of this research can be summarized as follows:
Flow-3D as an analytical model, due to the use of VOF method, has an appropriate capability in modeling free surface flows. RNG turbulence model, due to having additional term in the epsilon equation, compared to other turbulence models, is much more appropriate in analyzing quickly strained flows and the flows on surfaces with large curves. This model also has high potential to simulate transient flows.

1. In studying changes in static pressure of Azad dam's spillway, between the experimental and numerical data, there was little difference ranging from 3 to 5 percent. However, the greatest pressure difference was seen on the flip bucket, because due to high velocity in this point, there was turbulent flow.
2. Along the chute, due to significant reduction of pressure, there is the possibility of negative pressure and cavitation. Therefore, in order to the lower the risk of cavitation over the chute, at distances of 135, 185 and 245 meters from the threshold of the spillway, three aeration systems are recommended.
3. At a distance of 209.12 meters from the ogee spillway, maximum pressure reduction occurs. The reason for this reduction in pressure is that this point is the edge of the ramp in aeration system. However, the pressure is not in the range of negative pressure; therefore, at the presence of aeration systems, the likelihood of cavitation occurrence is reduced.
4. Pressure changes over the spillway of Azad dam are by and large affected by depth and velocity of the flow. With the increase in the flow depth, the pressure increased, and with the increase in velocity, the pressure reduced.
5. With the increase in discharge, flow velocity over the spillway increased.

6. References

1. Abdullah Kookhi, R. (2015), "*Numerical Study of Hydraulic Performance of the Spillway and Flip Bucket of Azad Dam*," Master's thesis, Islamic Azad University, Maragheh (in Persian).
2. Babazadeh, V, Manafipoor, M. and Mohammadi, M.A. (2012), "*Numerical Investigation of the Pattern of Flow over the Spillway and Flip Bucket of Gavshan Dam*," Ninth International Congress of Civil Engineering, Isfahan, Isfahan University of Technology.
3. Water Research Institute (Aaffiliated to the Ministry of Energy). (2010),"*Final Report of Hhydraulic Model of Azad Dam's Spillway*," September 2008.
4. Husseini Sahi, S.M. (2005), "*Hydraulic Simulation of Flow in Stilling Basin Using Flow-3D*," Master's thesis, Tarbiat Modarres University, Tehran (in Persian).
5. Ghasemzadeh, F. (2013), "*Simulation of Hydraulic Issues in Flow-3D*," Noavar Publications, Tehran, First Edition (in Persian).
6. F LOW 3D user manual, ver .9.3