American-Eurasian J. Agric. & Environ. Sci., 13 (3): 378-383, 2013
ISSN 1818-6769

DOI: 10.5829/idosi.aejaes.2013.13.03.458

Determine the Appropriate Location of Aerator System on Gotvandoliadam's Spillway Using Flow 3D

[1]*Mohammad-Hossein Erfanain-Azmoudeh* and [2]*Amir Abbas Kamanbedast*

[1]Islamic Ahvaz University-Ahvaz Branch, Iran
[2]Department of Water, Ahwaz Branch, Islamic Azad University, Ahwaz, Iran

Abstract: Cavitation is one of the most complex phenomena and the most common cause serious damage to spillways and chutes and is influenced by many factors such as pressure and flow rate, strength of materials, timeof operation and air flow rates. Conventionally, cavitation is controlled by the Cavitation number. Study a spillway to assess changes in cavitation number, need to check each model individually. Be time consuming and problems caused by changes in the size and the high cost of laboratory models and other difficulties on the one hand and the increasing spread of Computational Fluid Dynamics models and their capacity to on the other hand, engineers are interested in using this software. One of the most effective and most economical to avoid cavitation damage is ventilation in the stream. Thus, in this paper the FLOW-3D simulation is done to calculate the flow parameters to determine the location of the aeration systems of the chute. To simulate a turbulent flow, the standard equations are finite volume method and the method used fluid volume is used to determine the profile of the free surface. The results were validated with published data and good agreement was obtained.

Key words: Aerator System · Spillway · Chute · Cavitation · FLOW 3D

INTRODUCTION

An important factor that threatens stability spillways and chutes and it may damage irreparable these structures is the known phenomenon of cavitation. Until now, the cavitation damage has been discovered in large overflows in the world, such as Russia, Pakistan, Venezuela and especially Iran. In Iran, cavitation damaged a large portion of the Karun dam's spillway. On overflow, due to the roughness it is possible that the flow of water converges locally and consequently the velocity of the flow, when in a liquid animated high speeds, the local pressure decreases to become less than the saturation vapor pressure, is formed of small gas bubbles [1]. These bubbles can grow, coalesce, or collapse when a cavity driven in the flow through areas where the pressure increases again, it resolves abruptly (in a time of the order of milliseconds only), hence the name "collapse" given this phenomenon. At the end of collapse velocities and pressures in the immediate vicinity of the cavity walls are very large (pressures of 10 000 bar have been measured) then there is formation of a shock wave which, like any wave shock degenerates into a sound wave. The resulting noise is often the most obvious manifestation of cavitation is reduced if the cavity contains a sufficient amount of gas it, condensable, playing a role of mattresses [2]. The collapse of vapor cavities is also the origin of erosions that accompany cavitation. When a bubble shrinks, its shape is indeed unstable, there is a sort of micro-jet which, in the vicinity of a solid wall, tends to move towards it. Velocity in the jet is high (almost 100 m / s), it produces a micro crater when it hits the wall. When a solid surface and is the seat of repeated collapse, it starts to look orange peel and after stripping material, an aspect of sponge, before being pierced from side to side. The resistance of a material to cavitation erosion is naturally linked to the other mechanical characteristics of the material (resilience, hardness, in particular).When these bubbles reach a zone where the pressure is higher; they implode producing a shock wave. The latter induces a local fatigue of the material leading to repeated disruptions of the passive film and localized attack.

Corresponding Author: Mohammad-Hossein Erfanain-Azmoudeh, Islamic Ahvaz University-Ahvaz Branch, Iran.

Cavitation can occur in the case of a liquid at rest or in liquid flows, the concrete surface will receive strong impacts of these small jets and high pressure is created. This pressure can be enhanced by the waves produced by the same phenomenon and localized tensions come. Therefore, the concrete surface begins the phenomenon of fatigue and finally destruction [3]. Collision small jets with high speed concrete can produce a high dynamic pressure which is transmitted by localized capillary gaps and pores in concrete and may cause separation of a large piece of concrete structure.Because of the importance of protecting the structure spillway of dam, researchers in the last decades, tend to more accurate methods and more accurate methods are numerical methods. Simulate the flow over the spillway, before building the structure because researchers have a good view of flow on structure. In this research, the spillway-chute dam GotvandOlia was analyzed under the conditions varied by two-dimensional and three-dimensional simulation.

Since the early 1950s, the Army Corps of Engineers U.S. vessels Experiment Station studied the flow behavior of the spillway on using physical models. A series of design charts are available for hydraulic engineers profile spillway design for all levels given floods. The use of the technique in the analysis of CFD flow spillway is relatively recent. Difficulties involving initial solution convergence and moving grid to track the surface of the water have been reported. Nowadays, more efficient CFD codes can solve the Navier-Stokes equations in three dimensions and they also have a number of turbulence models to choose from. Define the geometry and three-dimensional mesh has been simplified, some codes can even transfer the geometry of the writing of others and / or programs of computer-aided design.In the design of the spillway profile is designed so that when water flows over the spillway structure under the maximum flow, it will not cause adverse effects such as cavitation at the crest and downstream. Ideally, the surface of the spillway must experience the atmospheric pressure in the head design [4].When the tank level is lower than flood level, the pressure above the spillway will be above atmospheric. When the tank level is above the head design, sub-atmospheric (negative gauge pressure) will take place along the crest of the spillway which can damage the surface of the concrete spillway due to cavitation and undesirable affects other components, including gate structures.

MATERIAL AND METHODS

Gotvand Olya Dam is located in the north of the city Shushtar, 240 meters above sea level. Model physical dam built in the Water Research Institute in Tehran and 1240 meters above sea level. So considering, 1000 meters level difference between the place of study and place of construction. Fig1.

Also Physical model of Gotvand dam's spillway-chute is shown in Fig 2.

In this paper the FLOW-3D simulation is done to calculate the flow parameters to determine the location of the aeration systems of the chute. To simulate a turbulent flow, the standard equations are finite volume method and the method used fluid volume is used to determine the profile of the free surface.The CFD code used for this study was FLOW-3D, which solves the Navier-Stokes equations by the finite difference method. The algorithm is an extension method based on the SOLA method which was developed at the Center Los Alamos National Laboratory Hirt et al. The volume of fluid is used to calculate the movement of free surface.All differential equations which govern such as continuity and the equations of motion are formulated with zones (2D) and volume (3D) functions of porosity. This formulation, Fractional Representation Obstacle Surface / Volume is used to model the complex geometric region.Any obstacle complex geometry can be represented using the technical support. The part of the volume (or area 2D) occupied by the obstacle in each cell (grid) is set at the beginning of the analysis. The fraction of fluid in each cell is also calculated. The continuity equation, equation of motion, or a transport equation of the fluid fraction is formulated by using the favor. A finite difference approximation is used for the discretization of each equation. Unlike other methods of elements / volume or limit the mounting grid finite, mesh technique does not require remising and would not be any distortion mesh in transient analysis where precision solution algorithm can be easily applied [4]. The basic algorithm for feeding a solution in an increment of time comprises the following three steps.

Step 1: Calculate the velocity using the initial conditions or previous time-step values for all advection, pressure and other accelerations based on approximations explicit momentum (Navier-Stokes) equations.


Fig. 1: GotvandOlya Dam


Fig. 2: Physical model of Gotvand dam's spillway-chute

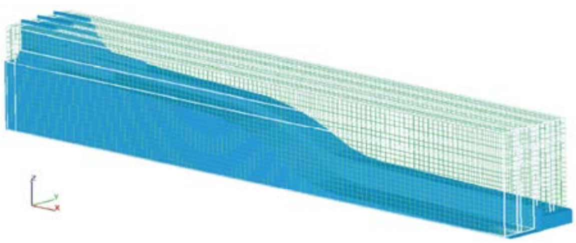
Fig. 3: Simulation model of Gotvand dam's spillway-chute

Step 2: Adjust the pressure to satisfy the continuity equation.

Step 3: Update the free fluid surface or interface to give the configuration of new fluid based on the volume of fluid.

For flow modeling using the software on the spillway-chute and we need to use one model for turbulent. In this research, the RNG by selecting the value of 0.07 as Turbulent Mixing length was chosen. In basic manual book FLOW-3D (Flow Science 2007), the RNG model is recommended as the most accurate model and the most powerful available in this software [5,7].

- Limit upstream hydrostatic pressure with zero speed, fluid height = H;
- Limit Downstream: An outflow boundary;
- Low Upstream: No flow - blocked by an obstacle below;
- Downstream Background: A flow limit;
- Upper: Symmetry - no effect in this case because of the gravity.

The baseline was set so that the volume of fluid with a head H is situated on the crest of the spillway. The transient flow analysis was performed for a total period of 100 seconds when steady state has been reached. This was determined by the inspection results such as speed and kinetic energy of the system. A density of 1000 kg/m3 constant water was used. This implies that the water is incompressible. Value of gravity 9.81 m/s2 was applied in the negative Z direction. To simulate eight speeds were chosen. (2000, 4000, 6000, 8000, 10000, 12000, 15000, 17000 cubic meters per second.). Figure 3 shows simulation model of Gotvand dam's spillway-chute.

Cavitation Number: Cavitation occurs depends on the pressure and speed of flow, which is shown by the Cavitation number. Office of the United States claims an paper under the title "Cavitation on spillways and chutes" (1990), presented the Cavitation Number as follows:1)

$$\sigma = \frac{P - Pv}{\rho V^2 / 2} \tag{1}$$

Where P (absolute pressure), Pv (vapor pressure) and V (flow velocity). One indicator that is used to determine the place of occurrence of cavitation on spillways and chute is the comparison between the Cavitation numberand Critical Cavitation Number [6]. The cavitation number is calculated by equation below, using pressure and speeds calculated or measured and any of the flow rates throughout the chute.

$$\sigma = \frac{\frac{Patm}{\gamma} + h\cos\alpha + \frac{hV^2}{gR} - \frac{Pv}{\gamma}}{\frac{V^2}{2g}} \tag{2}$$

Where, (the ambient pressure in the situation that is equal to a laboratory atmosphere, or 10.33 meter water column) (vapor pressure at 25°C which is equal to between 0.33 to 1 column of the mother water), or hcosè (pressure equivalent water height is measured in different parties of the structure) and (the equivalent height velocity in m / s section).In this equation the parameters (pressure difference resulting vertical arcs) is not considered.Substituting the values in the last equation, equation (2) is summarized and the results are as follows:

$$\sigma = \frac{(10.33 - 1 + \frac{P_0}{\gamma})}{V_0^2 / 2g} = \frac{2g}{V_0^2}(9.33 + \frac{P_0}{\gamma}) \quad (3)$$

The vapor pressure changed as the following:

$$\sigma = \frac{2g}{V_0^2}(9.74 + \frac{P_0}{\gamma}) \quad (4)$$

By putting the values of pressure and flow velocities of each section in the equation number 4, the cavitation number is obtained.By comparing the number of critical cavitation and cavitation number in the different sections, it is realized that cavitation may occur where the number of cavitation flow is less than the number of critical cavitation.Agreement studies, the value of the critical cavitation number structure is equal to 0.25.

RESULTS

Determining Location System Aerator: In this research the values of pressures and speeds for 9 sections (Table 1) were measured with FLOW-3D and using the equation number 4, the number of cavitation was calculated.It should be noted that the geometric properties of the chute and the chute left right are different, so the simulations were done separately for them.

Figures 4 to 11 show the values of the numbers of cavitation and spillway channel for different flow rates. The graphs show that in the range of 210-520 meter peak leads straight chute and 210-560 meter peak leads left channel, the cavitation number is under the critical line, it means the value of 0.25, then the occurrence of cavitation can design and implement systems and aerators in the ducts is required. As is clear from the curves of cavitation number, the first section, where the coefficient of cavitation is less than the critical value, ie, 0.25, is the horizontal distance of 210 m crest weir the left and right channels. So this place is considered as the building site of the first system in the aerator left and right channels.

Table 1: Measurement locations sections of spillway-chute

	Right Canal		Left Canal	
Section	Elevation (m.a.s.l)	Distance from Crest (m)	Elevation (m.a.s.l)	Distance from Crest (m)
E	218	0.00	218	0.00
F	204	28.45	204	28.45
G	198	124.24	198	124.24
H	194	196.59	194	196.56
I	176	253.45	182	241.06
J	146	292.25	156	284.35
K	130	332.26	144	338.37
L	126	424.03	140	452.65
M	124	549.28	140	552.45

Table 2: Locations of sections measurement spillway-chute

		Numerical Model	Physical Model
	Section	Distance from Crest (m)	Distance from Crest (m)
Right Chute	The 1th aerator	212	201
	The 2nd aerator	255	255
	The 3rd aerator	350	349
	The 4th aerator	400	410
Left Chute	The 1th aerator	212	201
	The 2nd aerator	255	255
	The 3rd aerator	370	380
	The 4th aerator	455	460

Profile of the first aerator system is shown in Figure [12]. The dimensions of the ramp were chosen by considering the dimensions of the ramp in the physical model.

To verify the location of the first aerator system and also determine the location of the vent system according to the values of the cavitation number in this area were also calculated and the length of jet flow for two of 12000 and 15000 cubic meters per second were measured. The reason for choosing the flow is still the cavitation number and, as the curves show, the cavitation number for both flow before others is less than the critical threshold. At this stage of the simulation, in order to accelerate the execution of simulation capacity nested mesh has been used in areas of ramps.Finally, simulating flow in the two channels when the ramp of the first system was built aerator, the downstream cavitation number was analyzed in the ramp area and downstream of it. After inserting a vent ramp in the direction of each channel and even simulate the flow, the location of the next ramp was determined by assessing the number of cavitation in the new state. Figure 13 shows the flow simulation on the first ramp.

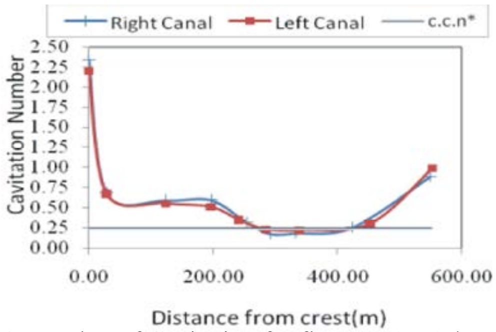

Fig. 4: Number of Cavitation for flow 2000 m3 / s

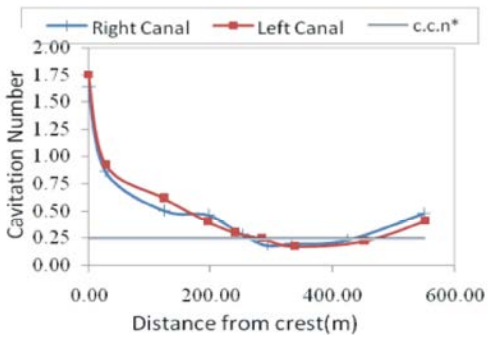

Fig. 5: Number of Cavitation for flow 4000 m3 / s

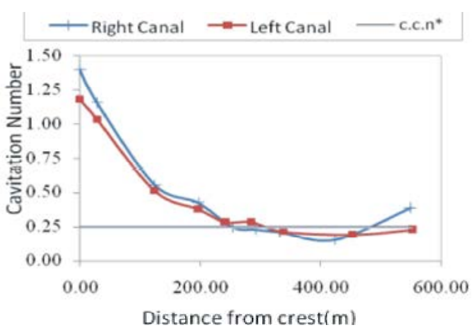

Fig. 6: Number of Cavitation for flow 6000 m3 / s

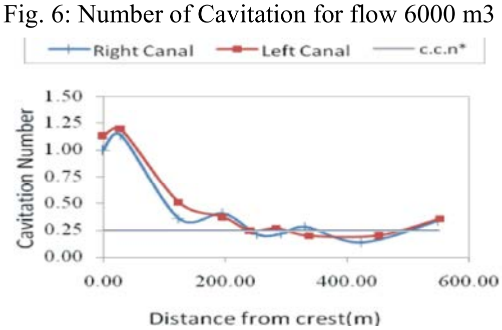

Fig. 7: Number of Cavitation for flow 8000 m3 / s

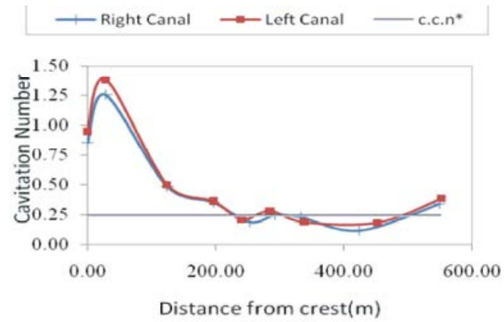

Fig. 8: Number of Cavitation for flow 10000 m3 / s

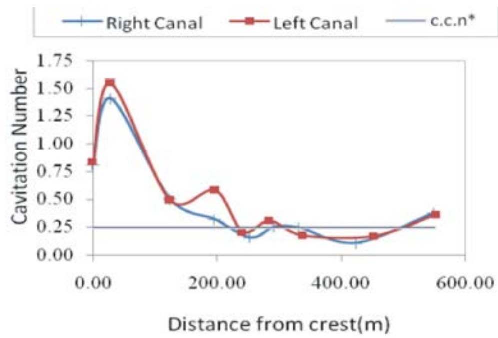

Fig. 9: Number of Cavitation for flow 12000 m3 / s

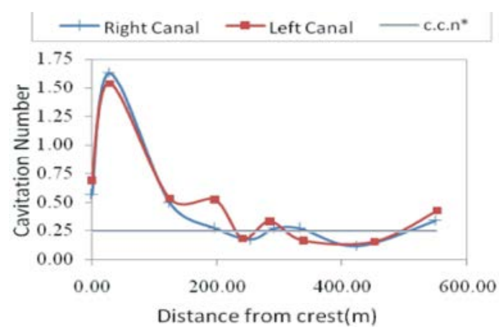

Fig. 10: Number of Cavitation for flow 15000 m3 / s

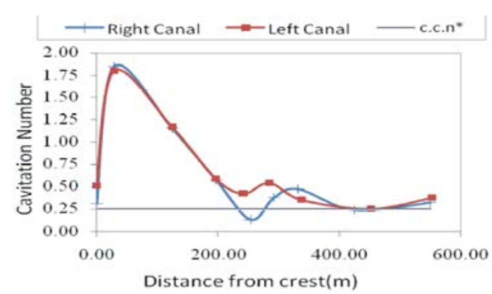

Fig. 11: Number of Cavitation for flow 17000 m3 / s

Validation: Validation results of cavitation number are calculated using FLOW 3D, with the results measured by the physical model, it was found that this results in a total average difference is 0.03% (Table 2).

After inserting each aerator ramp in the direction of each channel and evaluate their effects, it was found that four ramps for right channel and left channel four ramps are needed. Figures (14 and 15) show the results of cavitation number and the location of each aerator.

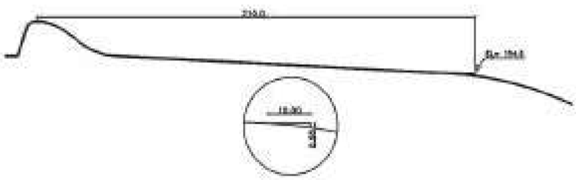

Fig. 12: Profile of the first ramp ofthe first aerator system

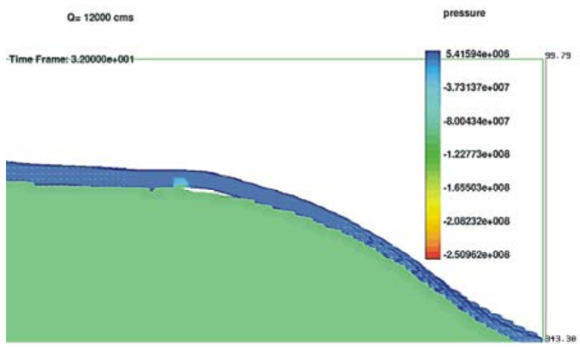

Fig. 13: Simulation of flow entitled chute flow 12000 (m3 / s)

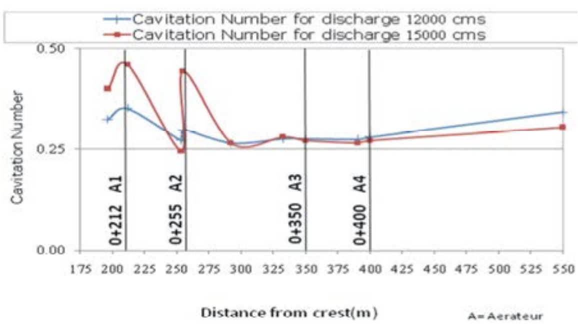

Fig. 14: The final evaluation of cavitation number after insertion ramps (Right bay)

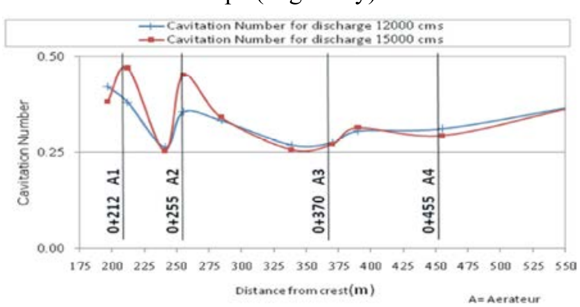

Fig. 15: The final evaluation of cavitation number after insertion ramps (Left bay)

DISCATION AND CONCLUSION

- Looking at the numbers cavitation curves (Figures 4-11), we find that in the area of distance 210-560 meter peak, the cavitation number is less than the value of 0.25 (the critical cavitation number) and the appearance of cavitation is possible. Then design and implement systems aerators in the ducts is required.
- Figures (4-11) also show that the minimum number of cavitation occurred in the right channel for flow of 12000 (m3 / s) (section L, $\sigma = 0.11$) and in the left channel for flow 15000 (m3 / s) (section L, $\sigma = 0.16$).
- Validation results of cavitation number are calculated using FLOW 3D, with the results measured by the physical model, it was found that this results in a total average difference is 0.03% (Table 2), which is acceptable and indicates that CFD can simulate well flow on spillways and chutes and can be used for the following projects.
- Final control after the insertion of four systems aerator on the right channel and the left channel shows that the number of aeration systems was sufficient and cavitation number is not less than the minimum value.

REFERENCES

1. Davis, C.V. and K.E. Sorensen, 1990. "Handbook of Applied Hydraulics", McGraw Hill.
2. ValiSamani, H., 1997. "Design of Hydraulic Structures" book publishing business consulting engineers Dézab. (In Persian)
3. Goharrizi Zandi, F. and M. Fathi Moghadam, 2010. "Investigation of changes Cavitation Number on spillway Model Using CFX" Fifth National Congress on Civil Engineering, Mashhad-IRAN. (In Persian)
4. Ho, D.K.H., K.M. Boyes, and S.M. Donhoo, 2001." Investigation of Spillway Behavior under Increased Maximum Flood by Computational Fluid Dynamics Technique" 14thAustralasian Fluid Mechanics Conference Adelaide University, Adelaide, Australia.
5. Flow Science, Inc. 2007. Flow-3D User's Manuals, Version 9.2, Santa Fe, NM.13- Guy 14-Chanel, P. 2008." An Evaluation of Computational Fluid Dynamics for Spillway Modeling".Master's thesis. Department of Civil Engineering University of Manitoba Winnipeg, Manitoba, Canada.
6. "Cavitation in chute and spillways", engineering monograph No.42, U.S. bureau of Reclamation,1990.
7. SamanAbbasi, Amir Abbas kamanbedast. 2012. "Investigation of effect of changes in dimension and hydraulic of stepped spillways for maximization energy dissipation"World Applied Sciences Journal, 18(2): 261-267.