

Aeration in Bottom Outlet Conduits of Dams for Prevention of Cavitation

Roya Kolachian^{1*}, Akram Abbaspour² and Farzin Salmasi²

¹M. Sc. Student of Civil Engineering, Islamic Azad University, Ahar Branch, Iran

²Department of Water Engineering, Faculty of Agriculture, Tabriz University, Tabriz, Iran

*Corresponding author's Email: roya.kolachian@yahoo.com

ABSTRACT: The more necessity of human communities to water resources demand leads to establishing and development of high dams and hydraulic structures with high capacities. With increasing in dam height, the water velocity in bottom outlet conduits increases. In the case of decreasing of local pressure due to sudden change in flow area or the passage slope the possibility of cavitation occurrence through the structure increases. Experiences showed that the flow aeration can decrease the risk of cavitation damages effectively. The investigations of flow characteristics by numerical methods along with physical hydraulic models can be effective in choosing the best choice for aeration and also decrease extra cost and time of physical modelling. In this research, first, the cavitation occurrence in bottom outlet of Sefidroud dam was evaluated in different gate openings by Fluent software in 3D mode and then after the comparison results of numerical and physical model, a method for aeration of this structure consist of aerator ramps on floor and side walls of conduit is suggested and investigated by numerical model. For this purpose the VOF model was used for flow simulation in tunnel and the water surface profile, mean velocity, static pressure and cavitation index were investigated before and after aeration. The results showed the relative capability of numerical models in flow simulation in bottom outlet conduit of dam and improving the flow conditions after aeration.

Keywords: Aeration, Bottom outlet conduit, Cavitation, Fluent,

ORIGINAL ARTICLE

INTRODUCTION

Bottom outlet conduits are the most important side constructions in dams which are used to control of reservoir water volume in back of dam, regulation of downstream water right and drain of the reservoir sediment (Najafi and Kavianpour, 2008). A main problem of these structures are cavitation phenomena specially after control gates which are occurred in different openings of gates in bottom outlet conduit of dams because of high flow velocity and pressure drop after gates.

Cavitation occurs whenever the pressure in the water flow falls to the value of the pressure of the standard water vapor. Cavities filled by vapor and partly by gasses excluded from the water as a result of the low pressure, are formed. When these bubbles are carried by the flow into regions of higher pressure, the vapor quickly condenses and the bubbles implode, the cavities being filled suddenly by the surrounding water. Not only is this process noisy, with disruption in the flow pattern, but more importantly if the cavity imploded against a surface, the violent impact of the water particles acting in quick succession at a very high pressure, if sustained over period of time, causes substantial damage to the concrete or steel surface, which can lead to a complete failure of the structure (Novak et al, 2004).

Cavitation in outlet conduits of dams is made many problems in Iran dams such as Sefidroud and Shahidabbaspour for a long year. Experiences showed

that air injection to flow downstream of the gate can dramatically decrease the risk of cavitation damages. Peterka (1953) used the aeration instruments for 2% air concentration and the cavitation damages are severely decreased in borders. He also found that 6-8% of air concentration can eliminate damages completely. After this research, the utilization of deflectors quickly increased in downstream of high dams for flow aeration (Hager et al., 2009).

The deflectors and ramps are currently the most common of aeration methods which are aerated with raising the flow in shoot or weir and free jet flow formation to lower the nappe. When the water flow get near the borders, the enough air is achieved, this can support the downstream surface of structure in against of cavitation damages (kolachian.2012).

According to Water Research Institute (2007), the characteristics of jet nappe on the ramp are as follows: primary velocity, the length of jet and maximum height of jet. In theory, the length of jet nappe on the ramp follows by below equation.

$$L_j = \frac{V^2}{2g} \sin 2\theta + 2 \cos \theta \left[\frac{V^2}{2g} \left(\frac{V^2}{2g} \right) \sin^2 \theta + h \right]^{1/2} \quad (1)$$

Where L_j is the jet length from ramp end to place of jet reclusion to bottom of tunnel, V is flow velocity, h

is ramp height and θ is the ramp angular to horizon. h_j is the highest of jet height and calculated with this equation:

$$h_j = \left(\frac{V^2}{2g}\right) \sin^2 \theta \quad (2)$$

The appropriate selection of ramp height and angles make possible the jet spraying in desirable place in the downstream of shooting and prevent the jet collision to tunnel ceiling. In addition, the suitable air vent design in tunnel walls can transfer air from the top of water surface or outside of tunnel to the under of flow nappe.

The air is getting sucked to water flow by bottom aeration and the creation of negative pressure in bottom of jet flow. The mixture of water and air absorbed the created impact due to bubble explosion. Air presence increases the exchange of atmospheric gases such as nitrogen, oxygen and carbon dioxide with the air-water mixture and the compressibility of flow water decrease damages on concrete or steel surface of structure (Shamsai and Soleymanzade, 2009).

MATERIALS AND METHODS

The studied Sefidroud Dam is located near Manjil city in Guilan province in north of Iran and has built on Ghezeloan and Shahorud rivers conflux. This dam is concrete gravity with height of 106 m and length of 425 m. The total volume of dam reservoir is 1.8 milliard m^3 which it has 1.2 milliard m^3 as the useful volume of dam reservoir. The level of Sefidroud dam crest is 277.06 m from the sea level (Water Research Institute, 2007).



Figure 1. Schematic of Sefidroud dam

According to being erosion catchment of Sefidroud dam, the useful volume of dam reservoir is quickly decreased because of sedimentation in primary years of operation. One of the sediment evacuations in dam reservoir is the sediment flushing method by bottom discharge gates in free or under pressure flow state with high velocity. After several flushing operations, some damages are created in different sections of steel covering gate, the steel covering of transition gallery in downstream of gate and concrete coating floor and gallery walls (Water Research Institute, 2007). Some examples of these damages are showed in Fig 2.

The studying and solving of arisen problems in bottom outlet conduit are performed with physical model studies. For this purpose, a model was build similar to

real structure with smaller dimensions in hydraulic laboratory of Tehran water research institute. The hydraulic characteristics of structure are studied by dynamic similarity creation between model and main prototype and the results of these studies have been used for evaluation numerical model verification.



Figure 2. Damages in Sefidroud dam

Ramps locating:

The results of data analysis showed that the water flow separation from right wall has been occurred in 15 m downstream of gate and the negative pressures are so dangerous because of cavitation so, aerator ramp should be installed in appropriate place for flow aeration and destroy the negative pressure region. At a distance of 7 meters after gates, tunnel has an opened section which tunnel height increase from 3.3 to 6.6 m.

The Ramp produces a jet which goes through parabolic path. If the aerator ramp is installed between gate and section of tunnel with height increasing, for more gate openings the flow jet will knock up to tunnel ceiling and produced complex hydraulic condition. This subject has caused that the place of ramp installation should be located after increased tunnel height. According two stated cases, the installation place of the ramp is at distance of 7.5 m from gate and 70 cm from increased tunnel height.

The studies of physical model showed that 50 m length of tunnel is faced with high velocity flow and negative pressures so, the aim of this project is the design of ramp on floor which can create a jet with 50 m length. In this regard, the length of jet is selected $L_j=50$ m using Equation 1, the ramp with angle of 11° and length of 2 m for floor aeration is designed. This kind of ramp create a jet with maximize height of 3 m and velocity of 40 m/s that the flow will not knock up to ceiling because of tunnel ceiling with 6.6 m height. The ramp installation on the tunnel floor produce jet flow and the spaces under jet are faced with negative pressures. To establish a connection between the spaces under jet to atmosphere, two ramps are created on the side walls. According to the criteria stated by Falvey (1990), the dimension of wall ramps should be equal or higher than 1/12 width of flow, also it should not be less than 100 mm. By assumption length of wall ramps equal to 2 m, the angle of 7° is selected as a walls ramp angle which has 240 mm height. This ramp conform the suggested criteria by Falvey (1990). The average width is 250 mm (1/12 of section width equal to 3 m). The height of ramp walls is chosen 4 m to support maximum flow depth. Ventilator duct is located on tunnel ceiling after gate and

open end of tunnel is used for required air of jet flow. Ramps locating is shown in Fig. 3.

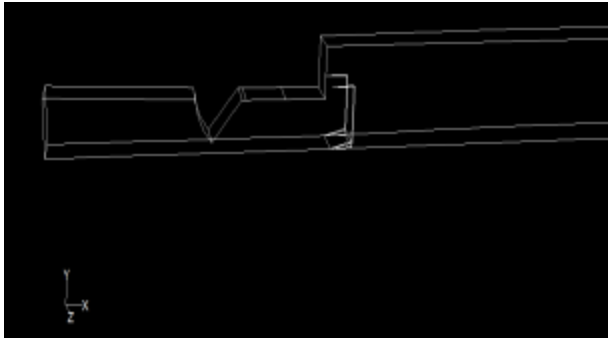


Figure 3. Ramps location in the conduit

Numerical modelling:

Numerical modelling has three stages. The First stage is design of structure geometry by Gambit.

To use CFD software it is firstly required to prepare the structure geometry using mesh generator software and then import to CFD media. For this purpose Gambit software is used for mesh file production and the prepared mesh file is performed in Fluent program with .msh suffix. In this regard, three-dimensional model of bottom outlet conduit for Sefidroud Dam is prepared using available maps from plan and side view of conduit in Gambit software. Geometry design of structure got started with more than 100 points coordination in duct length with scale of 1:15. Then the geometric model is drawn in real scale with three dimensional configuration using Gambit software. The vertex points are created at first. Then edges, faces and volumes are developed by vertex, edges and faces connection and combination, respectively. Finally the volume including main volume of duct, gate, ramps of walls and floor prepared as a general model of conduit for Meshing. The appropriate variation in this general model allows investigation of three type openings of gate with 20%, 60% and 95%. A combination of triangular grids is used for meshing of this model because of deformations in side walls and its floor and also the complicated geometry. The optimized mesh numbers are selected 17000 in each model which produce high accuracy answers and these numbers have been increased to more than 31000 meshes by adopting models in Fluent. By considering these numbers of triangular mesh in 3Dmodel, the convergence of model is in the range of 100-200 seconds.

It is noteworthy that the length of modeling is 141 m and its width is varied between 2.5 to 3.5 m and the tunnel height is 3.3 m before gate and 6.6 m after that (Fig 4).

The second stage is the simulation in Fluent. In the free surface flows, VOF model can be appropriate for investigation of vortex flow, air bulbs in water and fluid dynamic in dam breaking. VOF method can model two or several immiscible liquids using by momentum equations and volume fraction of fluid in computational domain. According to previous experiences in modeling of this conduit and also the occurrence of air and water interaction in tunnel length and during aeration process, the simulations are done by VOF model in the multiphase state. Also, standard $k-\varepsilon$ model is used for modeling of turbulence flow by comparison of different

methods to present the better results. For evaluation of gravity effect on flow, the open channel flow is used and the gravity acceleration is assumed 9.81 m/s^2 .

In Solver menu, the segregated equations methods are used. That is solved in three dimensional spaces and the equations are solved as implicit and unsteady state. VOF model is used for identification of water surface, it is recommended that the compressible fluid is introduced as an initial phase and the secondary phase is the fluid with a volume fraction of 1. For this reason, in this project air and water are introduced as initial (fluid 1) and secondary (fluid 2) phases, respectively. The model inputs are water and air pressure which is identified as pressure inlet and the model output is defined pressure outlet. Tunnel walls and gate are defined as the wall. Figure 4 shows the boundary conditions of the numerical model.

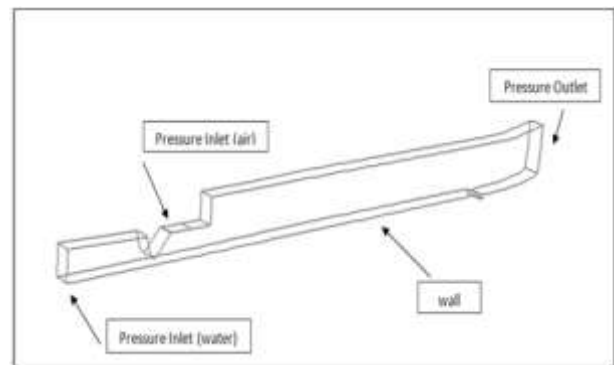


Figure 4. Boundary conditions of the numerical model

For coupling velocity-pressure scheme, the different equations are used. In present study PISO algorithm is selected because of the higher degree of the approximate, relation between the corrections for pressure and velocity and quickly convergence. For pressure interpolation scheme PRESTO! is used. It is recommended for flows with high swirl and for the VOF. The Second Order Upwind or Quick discretization scheme interpolates the face value of a variable with higher-order accuracy. The VOF at cell faces are computed with Quick discretization scheme in this research. This method can improve the answer accuracy. The First Order Upwind methods are selected for other equations in solution menu.

In this research, the time step is determined 1×10^{-4} and the repetition numbers in every time step is 20. The absolute convergence criteria for the residuals are selected 1×10^{-3} in all equations.

RESULTS AND DISCUSSION

For studying of flow characteristics such as velocity and static pressure and also the comparison of these values with results of physical model in specified distances of tunnel, some plates are drawn perpendicular to flow direction and the values of velocity and static pressure are defined in these sections. The contours of velocity and pressure are used for investigation of parameters variations before and after aeration.

Figures 5 and 6 shows the variation of air percentage because of aeration for 60% opening gate. As can be seen in these contours, water enters from the

tunnel entrance and progresses to end of tunnel. The required air of flow is prepared by aerator duct through top of gate and end of opening tunnel. Blue colour lighting after aerator ramps shows the increasing of air concentration in water after aeration.

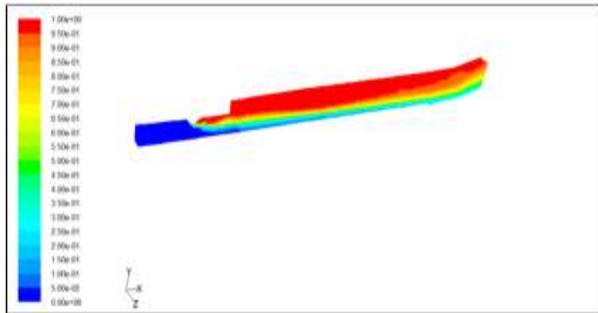


Figure 5. Contours of volume fraction before aeration

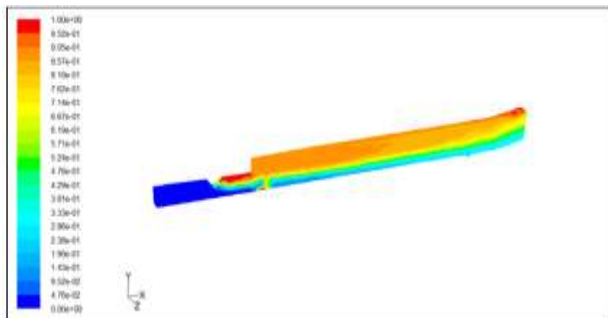


Figure 6. Contours of volume fraction after aeration

Cavitation Index:

The occurrence of cavitation phenomena is depending on pressure and flow rate which is expressed as a cavitations' number (cavitations' index) as follows:

$$\sigma = \frac{P_0 - P_v}{\frac{1}{2} \rho V_0^2} \quad (3)$$

Where P is absolute pressure, P_v is the vapor pressure and V is the flow velocity.

If the atmosphere pressure is 10.3 m, then the above equation is written:

$$\sigma = \frac{P/\gamma + P_{atm}/\gamma - 0.123}{V^2/2g} \quad (4)$$

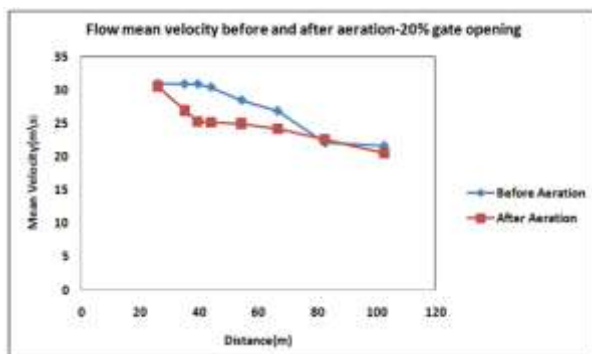


Figure 7. Mean velocity for 20% gate opening

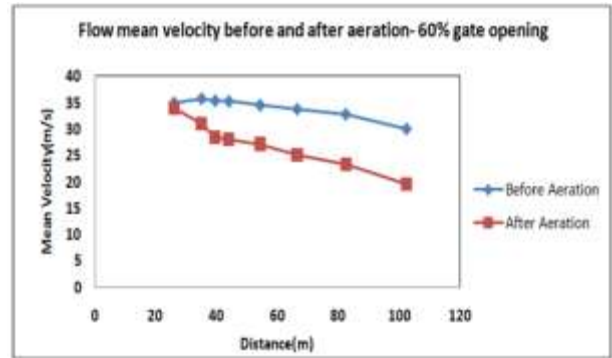


Figure 8. Mean velocity for 60% gate opening

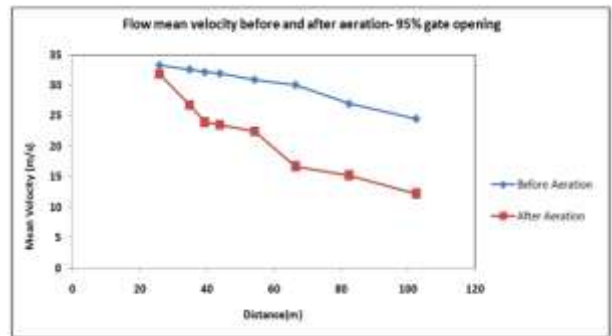


Figure 9. Mean velocity for 95% gate opening

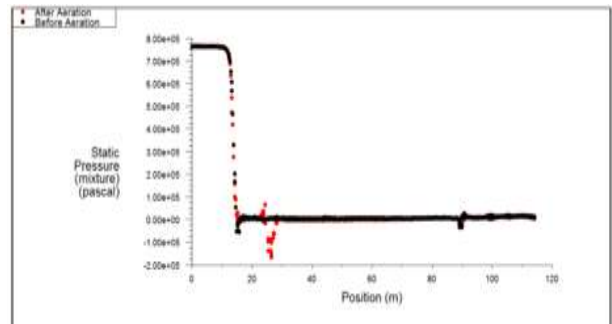


Figure 10. Static pressure for 20% gate opening

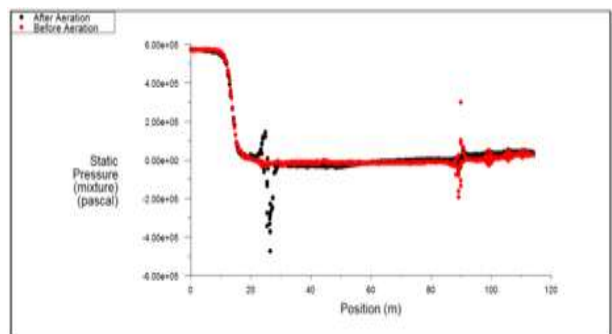


Figure 11. Static pressure for 60% gate opening

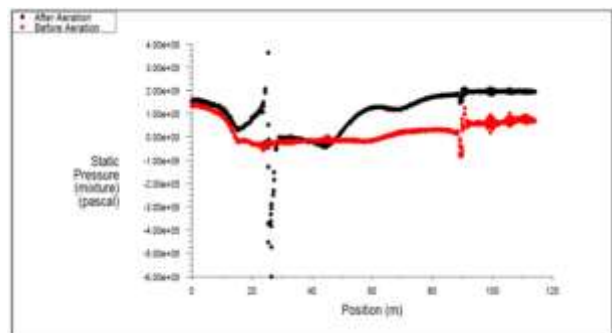


Figure 12. Static pressure for 95% gate opening

bottom outlet gates, 7th Iran Hydraulic conference, Shahid Abbaspour University, Tehran, Iran.

7. Novak P, Moffat A, Nalluri C and Narayanan R. (2004). Hydraulic structures, Third Edition, Spon press, Tailor & Francis e- library, New York.
8. Peterka A J. (1953). The effect of entrained air on cavitation pitting. 5th IAHR Congress Minneapolis, 507-518.
9. Shamsai A, Soleymanzade R. (2006). Numerical simulation of air-water flow in bottom outlet. International Journal of Civil Engineering. Vol.4, No.1.pp. 14-33.