Bulletin of Environment, Pharmacology and Life Sciences
Bull. Env. Pharmacol. Life Sci., Vol 3 (4) March 2014: 127-133
©2014 Academy for Environment and Life Sciences, India
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

Corrosion Evaluation of the cup-shaped Projectiles using numerical overflow dam Flow3D

Peiman Eskafi and Moslem Mansouri Nezhad
Department of Civil, Behbahan Branch, Islamic Azad University, Behbahan, Iran
Department of Civil, Behbahan Branch, Islamic Azad University, Behbahan, Iran

ABSTRACT
Usually when the flow rate of the hydraulic part of the structure of a limit beyond which structural damage caused by exposure to vacuum generation (cavitations) placed. Bumps and unevenness of concrete surfaces divert flow lines and reduce pressure in some areas If this is the result of pressure reduction is increasing rapidly, creating vacuum conditions or the onset threshold (corrosion) will come up [1].
In this research, using a strong software Flow3D model in the field of CFD, the High Dam overflow model and simulation on hydraulic flow and the important parameters of the corrosion phenomenon is investigated. For this purpose, overflow during the 16 stations were used to measure these parameters. Based on the results of measurements taken and given Flow3D model critical cavitations index (0.25), the probability of occurrence of cavitations phenomenon in the headwaters of the projectile does that the results are very consistent physical model built in the laboratory.
Keywords: cavitations, Flow3D, numerical simulations, the projectile cup.

Received 20 /12/2013 Accepted 09/02/2014 *©2014 AELS, INDIA*

INTRODUCTION

Effect of reducing pressure on evaporation process basically localized structures in constant temperature, corrosion or kavitasion is called. The bumps are and the smoothness of the concrete makes the diversion lines flow and pressure losses in some parts of that if it is in effect increase the speed of pressure decrease corrosion conditions arise.

In this case the pressure in the area of the lower vapor pressure and secretes a fluid which is part of the current state of boiling steam and turned to fluid and bubbles of steam are created. This is the path after the bubble during a short text to a higher level the regional pressure and explodes and blows the creation of waves and to the border between the fluid and the structure and impact on the solid border, shortly after the creation of erosion and corrosion. The conversion of pressure into the liquid and bubbles again due to blast it sometimes to 1000 mega Pascal.

Since the surfaces of the bubbles are too small for the bed overflow extraordinary force of the blast leads to concrete substrates. This can be done in a short time and with repetition, which causes corrosion of the substrate spillway, stilling basin bullet cup or structures to be some large holes will be gradually transformed into the fractures.

To prevent such injuries, it should be possible to create a specific design speed of corrosion or physical modeling in the meantime run by numerical modeling and studied to absolute roughness with detailed estimates of the required amount of speed has gotten.

To prevent cavitation phenomena in areas where they may be positioned speed increases, the pressure decreases as the vapor pressure could be identified. Therefore, to obtain a quantitative measure , the energy equation (Bernoulli's equation) between two points in a constant flow can be written :

$$\frac{\rho V_o^2}{2} + P_o + Z_o \rho g = \frac{\rho V^2}{2} + P + Z\rho g$$

P that pressure, P0 initial pressure, V velocity, V0 initial velocity, Z height, Z0 base balance, g ρ gravity and the density of water. (The value is zero, the upstream flow conditions). Energy equation in dimensionless form can be presented as follows:

$$\frac{(P+\rho gZ)-(P_o+\rho gZ_o)}{\frac{1}{2}\rho V^2}=1-\left(\frac{V}{V_o}\right)^2$$

Often the cause of equality Z0 and Z levels or low gravity effect of the sentence on the left side of the equation that matter, we have:

$$C_P=\frac{P-P_o}{\frac{1}{2}\rho V_o^2}$$

The pressure coefficient Cp as well as pressure parameters or Euler number Euler Number say.

Also, instead of replacing the symbol Cp P liquid vapor pressure at ambient temperature, Pv will be shown :

$$\sigma=\frac{P_o-P_v}{\frac{1}{2}\rho V_o^2}$$

That factor is called cavitation. Thus σ than the pressure drop required to evaporate the water pressure potential energy is flowing through . This equation can overflow the free surface with respect to the vertical curve at the bottom, novel as follows :

$$\sigma=\frac{\frac{P_{Am}}{\gamma}-\frac{P_v}{\gamma}+hCos(\theta)\pm(\frac{h}{g}\times\frac{V_o^2}{r})}{\frac{V_o^2}{2g}}$$

Where the density of water γ , θ steep angle of the bottom water to the horizon, r radius of vertical curve (symbol - the curvature of the curved concave and convex pluses) is .

Cavitation occurs when the σ σcr cavitation is equal to or smaller than the critical factor. this critical value mainly to the geometry, shape and surface roughness height and position datum point at which pressure and velocity have been measured, it depends.Based on the results of experiments of models, diagrams and semi-empirical relationships to determine σcr different conditions are provided.

According to the conventional facilities, the implementation of concrete hydraulic structures, often slope surface roughness at 1:20 overflow during the critical factor of 0.2 from the corresponding 0/25.(the slope against the direction of flow) and 0.25 (slope agreed to the flow direction), respectively. Figure 2: cavitations coefficient curve slope, roughness and against the flow of the two is shown. σ If this is less than the risk of cavitations is certain. Designers to identify potential cavitation in hydraulic structures, water surface profiles, or different discharge currents are calculated .Given the depth and average velocity, cavitations coefficient calculated at different positions σcr , and the values are compared to those places. In each section the σcr σ ,> the risk of cavitation that range there. Cavitation calculations must be performed for every different flow , critical because the state does not happen every flow. [1].

Introducing the model:

Flow3D model is an appropriate model for solving complex problems in fluid dynamics and is able to model a wide range of fluid flow. The software for modeling unsteady three-dimensional free surface flows with complex geometries has many applications. The application of the method, the volume of fluid (VOF) for solving the governing equations of flow in vertical gridding is used regularly. The discrete equations in finite difference method are similar to the discrete equations. Accordingly, the first and second order accuracy in software Flow3D method uses equations.

Accordingly, the first and second order accuracy in software Flow3D method uses equations. This software is applicable for five Prandtl mixing length turbulence model methods, a single-equation model, two equation k-ε, k-ε RNG two-equation Great Vortex Simulation (LES) to model and applying computational flow has not structured and solving optimization and network flows, compressible and non-compressible, viscous flows, laminar and turbulent flow modeling on various overflow, shallow water modeling, modeling of sediment in rivers, streams, biphasic, although.

Flow3D software solves the following energy equation:

$$\frac{\partial}{\partial t}(\rho E)+\frac{\partial}{\partial x_i}(u_i(\rho E+p))=\frac{\partial}{\partial x_i}\left(k_{eff}\frac{\partial T}{\partial x_i}-\sum_j h_j J_j+u_j(\tau_{ij})_{eff}\right)+S_h$$

k_{eff} : Effective conductivity (k + kt is the thermal conductivity kt turbulence, the turbulence model is used).

Jj ': Broadcast flux of species j' is. The first three terms on the right-hand side of equation (6), respectively, expressed as heat energy transmission, distribution and dispersal of species is viscosity. Sh: Includes heat volumetric thermal chemical reactions and other terms that users encounter will be.
Mass conservation or continuity equation, the incompressible flow can be written as follows:

$$\tau_{ij} = \left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] - \frac{2}{3}\mu\frac{\partial u_l}{\partial x_l}\delta_{ij}$$

Where μ is the molecular viscosity and the second sentence is right on the effect of volume expansion.
Method RNG, which is one of the newest models of turbulence using a complex statistical method based on instantaneous Stokes equations is obtained.
The form of the k-ε model has many similarities with standard methods. RNG theory to calculate the turbulent Prandtl number gives an analytical relationship, whereas a fixed numerical standard model is introduced.
The RNG model is more reliable and accurate results in a wide range of flows than the standard model offers. Here RNG method is used in the calculation of viscosity equations. [2]

Profile vitro model:
Model overflow dam in the headwaters of Water Science Engineering laboratory physical model has been constructed in 1:40 scale.
To perform the experiments, overflow to the profile width 50 cm, length 9 m and a height of 50 m cm were made and hydraulic parameters such as velocity and depth of water and the pressure and the flow rate measured by the cavitation phenomenon The projectile cup (Fig. 1) has been studied.

Figure 1 - Projectile cup vitro model headwaters dam spillway

FLOW3D the headwaters dam spillway was modeled using the model. This procedure is done in the software, enter the required data such as the shape and geometry of the structure and set of calculations to determine the execution time of a physical model, the flow phase, gender fluid mesh (the most important parts of the software Flow3D software and networking to the task of defining the configuration and the basic equations of fluid dynamics is responsible) and the physical boundary conditions governing the flow roughness coefficients, and, ultimately software and the model output control simulation type, interval, output range and output file size, will give us the information needed to evaluate the data obtained with the hydraulic flow on the projectile would overflow the cup.
Sample of overflowed (laboratory scale) input to the application, is as follows. (Figure 2)

 ©2014 AELS, INDIA

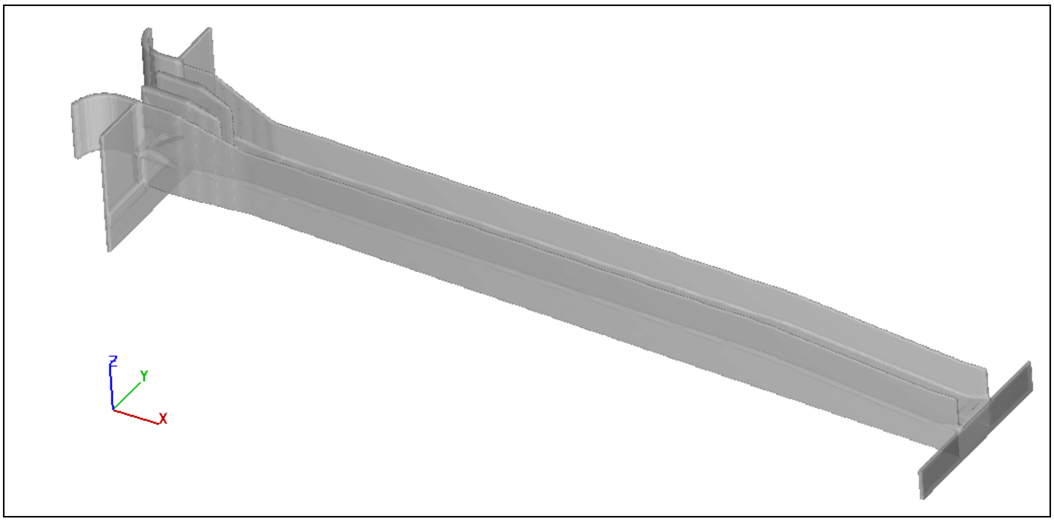


Figure 2 - The input overflow FLOW3D software and place the cup-shaped projectile

Examinations and measurements:

After entering the required information, software simulations have been conducted and the information required for the review will create a vacuum. In this study of four flow (66.7, 160, 193 and 378.6 liters per second) was used to perform the simulations. An example of flow simulations in Figure 5, 6 and 7 can be seen.

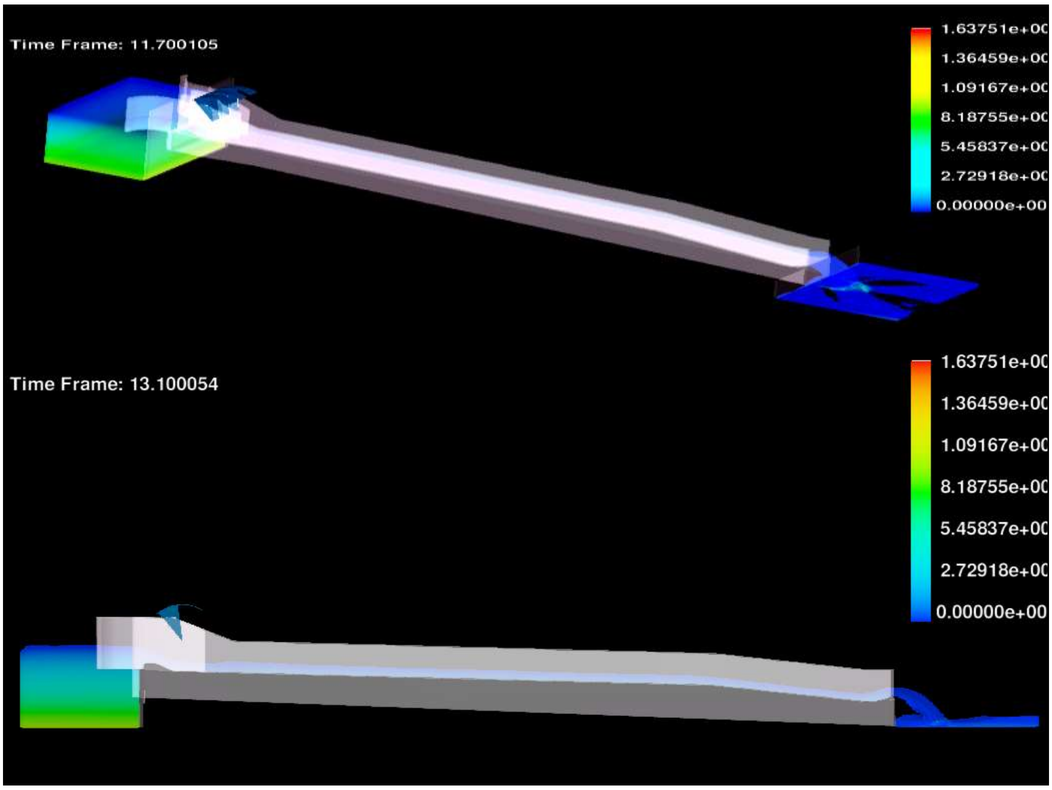


Figure 3 - Three-dimensional simulation of the flow over the spillway with a rate of 66.7 liters per second

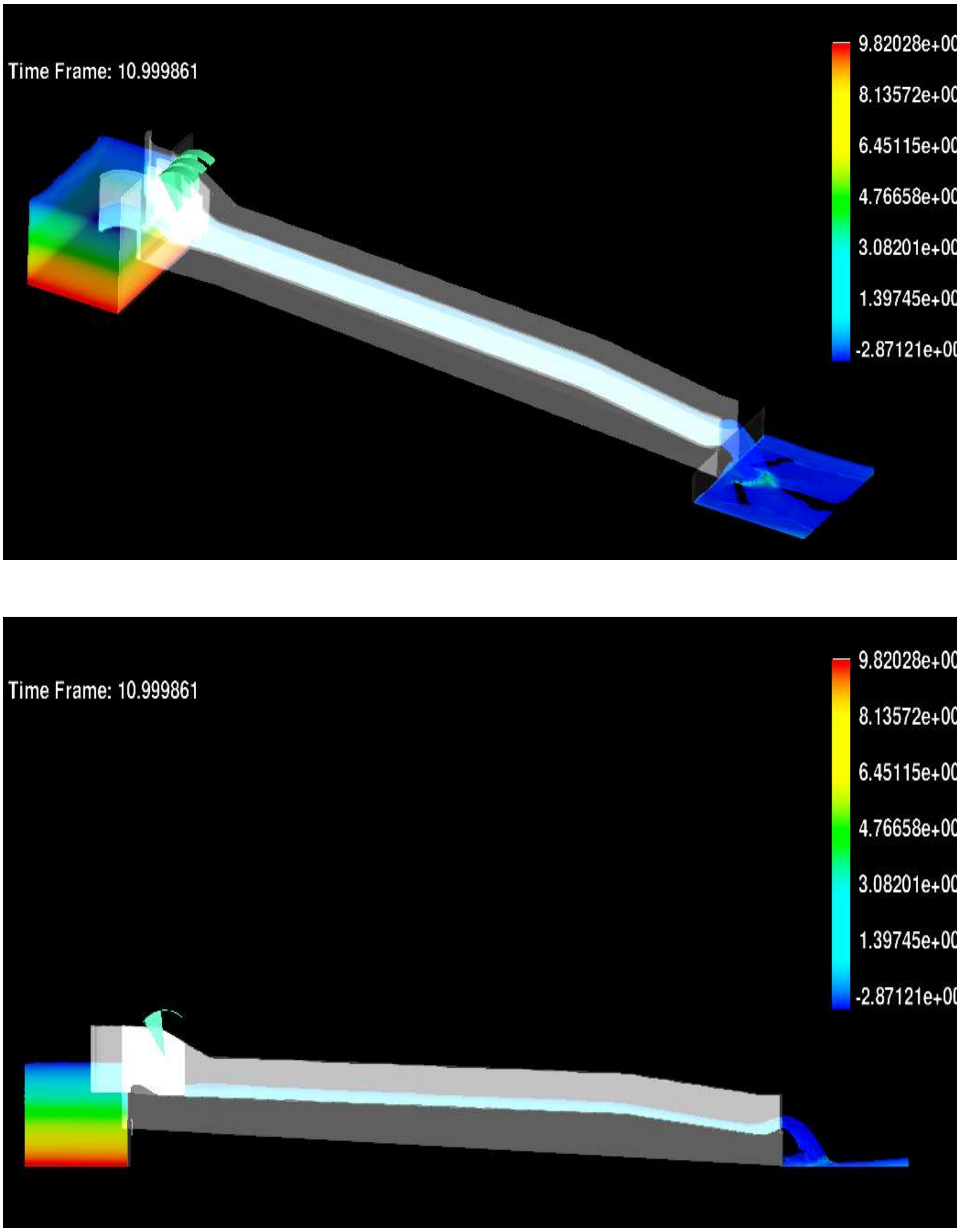
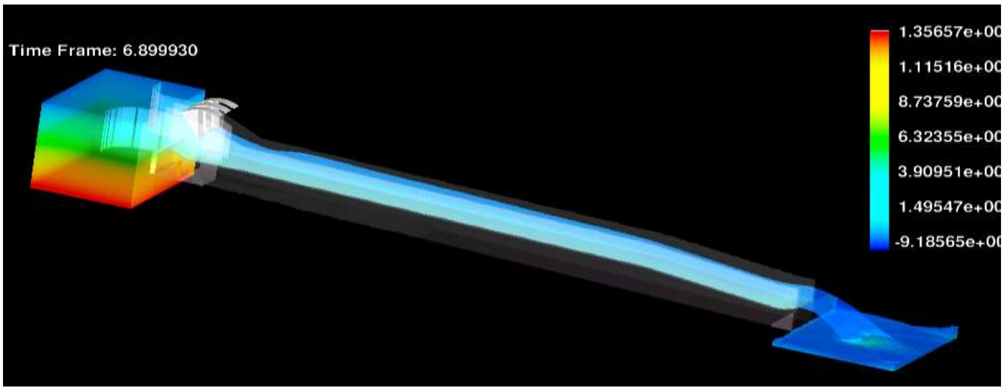


Figure 4 - Three-dimensional simulation of the flow over the spillway flow of 160 liters per second



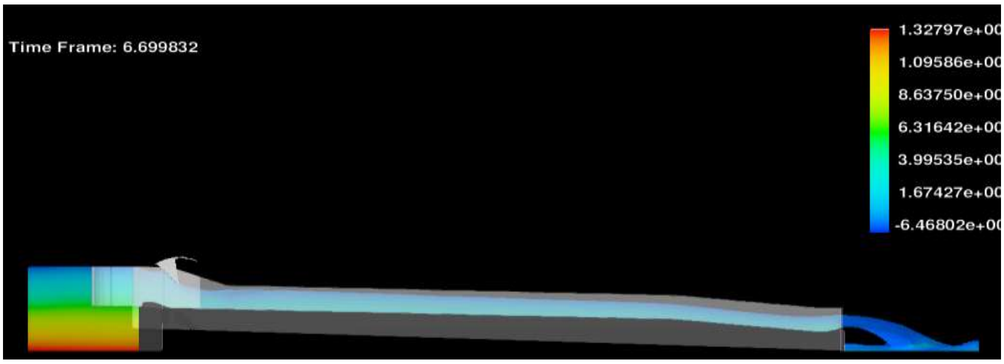


Figure 5 - Three-dimensional simulation of the flow over the spillway with a rate of 378.6 liters per second

After experimenting with the software FLOW3D and according to the above on how to calculate corrosion, cup-shaped projectile point of overflow, the data in Table 16 below spillway crest distance of any point of the listed data were extracted. It should be noted that the data points selected for this study because it is an in vitro model of the cup-shaped projectile points used to calculate the corrosion phenomenon..

RESULTS AND DISCUSSION

According to data obtained from the model and plotted graphs can be concluded that the lowest corrosion rate is 0.5 times the rate of 378.6 liters per second. Given the critical corrosion index 0.25, reduced risk of cavitation. By comparing the experimental and model data model FLOW3D can be concluded that this software can be used in hydraulic calculations and acceptable results can be harvested. So given the above, it does not seem necessary that the structure be used, such as aerators.

REFERENCES

1. Arif M, M., (1388), Investigation of Hydro Dynamics Trazhktvry jet stream and downstream of the projectile cup-shaped curve (case study dam spillway headwaters)", MA thesis, University of martyr Chamran.
2. Solar, c. Bidakhti, n. Nikseresht, AZ, (1387), Investigation of cavitations Drtkhlyh help lower White River Dam software - software FLUENT", Fourth National Congress on Civil Engineering, Tehran University, May 1387.
3. Morshedi, p., (1378), Study of cavitations phenomena on the overflow number of dams in Iran using computer models," Master's thesis, University of martyr Chamran.
4. M, M. (In 1385). Hydraulic flow behavior above the dam spillway is a small scale Master's thesis, martyr Chamran University.
5. Fazelipour, no. (1390). "Flows over the dam spillway hydraulic simulation using software Flow3D" Master's thesis, Department of Water Science, martyr Chamran University.
6. H.O, D., Boyes, K. and Donohoo, S. (2001). Investigation of Spillway Behavior under Increased Maximum Flood by Computational Fluid Dynamics Technique.14th Australasian Fluid Mechanics Conference, Adelaide University, Adelaide, Australia.
7. H,O, D., Boyes, K. and Donohoo, S. and Cooper, B. (2003).Numerical Flow Analyses for Spillways . 43rd ANCOLD Conference Hobart, Tasmania, 24-29 October 2003 .
8. Ikegawa M., Washizu K. (2005). Finite element method applied to analysis of flow over a spillway crest". International Journal for Numerical Methods in Engineering, Volume 6, Issue 2, Pages 179 – 189.

9. Kim, S. and Lee, H. and an, S. (2010). Improvement of hydraulic stability for spillway using CFD model ". J. of the Physical Sciences Vol. 5(6), pp. 774-780.
10. Sabbagh-Yazdi, s.a.,Rostami .F .(2007). Numerical Investigation of Pressure Load over Spillway Chutes with Vertical Curvatures ". 5th IASME / WSEAS International Conference on Fluid Mechanics and Aerodynamics, Athens, Greece, August 25-27, 2007.

How to cite this article:
Peiman E. and Moslem M. N. Corrosion Evaluation of the cup-shaped Projectiles using numerical overflow dam Flow3D.Bull. Env.Pharmacol. Life Sci. 3 (4) 2014: 127-133