

# Numerical Simulation of Cavitation and Aeration in Discharge Gated Tunnel of a Dam Based on the VOF Method

Razieh Jalalabadi, Norouz Mohammad Nouri

**Abstract**—Cavitation, usually known as a destructive phenomenon, involves turbulent unsteady two-phase flow. Having such features, cavitating flows have been turned to a challenging topic in numerical studies and many researches are being done for better understanding of bubbly flows and proposing solutions to reduce its consequent destructive effects. Aeration may be regarded as an effective protection against cavitation erosion in many hydraulic structures, like gated tunnels. The paper concerns numerical simulation of flow in discharge gated tunnel of a dam using RNG  $k - \varepsilon$  model coupled with the volume of fluid (VOF) method and the zone which is susceptible of cavitation inception in the tunnel is predicted. In the second step, a vent is considered in the mentioned zone for aeration and the numerical simulation is done again to study the effects of aeration. The results show that aeration is an impressively useful method to exclude cavitation in mentioned tunnels.

**Keywords**—Aeration, Cavitation, Two-phase flow, Turbulent Flow, Volume of Fluid (VOF) method.

## I. INTRODUCTION

SINCE many natural phenomena which relates to fluid dynamics concern with two phase flow, such flows have received growing research attention among fluid dynamics researchers [1]. Cavitating flows are a group of two phase flows which happens when there is a drastic pressure drop in fluid (usually due to high velocity) insofar as it falls to below the local vapor pressure of the water. So vapor bubbles will form and move downstream into a region where the pressure would increase and vapor bubbles collapse. This phenomenon causes destructive effects such as erosion, noise and vibrations [2].

Although in some cases cavitating flows are useful, like supercavitation; when the length of the body is less than the cavity length and is useful in drag reduction [3], cavitation causes destruction in many systems like pumps, valves and hydraulic systems. As it is expensive and time consuming to repair the damaged systems [4], it is important to predict the zones in which cavitation incepts, and use some schemes to prevent this phenomenon. Inasmuch as experimental and analytical methods applied by scientists to predict cavitation

inception are of high costs and impossible to be done for many systems, numerical schemes have been employed to simulate cavity flows and have led to better understanding of the dynamics of bubbly flows.

Two current approaches available for the numerical calculation of multiphase flows are the Euler-Lagrange approach and the Euler-Euler approach. The volume of fluid (VOF), the mixture model, and Eulerian model are three different Euler-Euler multiphase models which are usually used to model bubbly flows (usually in commercial softwares like ANSYS/FLOTARN) [5]. VOF method is used when the position of the interface between two fluids available in the domain is important. In this model a single set of momentum equations is solved for the fluids and the volume fraction of the fluids available in the domain is determined in each computational cell. Applications of the VOF model include free-surface flows, the motion of large bubbles in a liquid and the motion of liquid after a dam break [6], [7], [8]. In mixture model, the mixture momentum equations are solved and relative velocities are determined for available phases. The applications of this model include particle-laden flows with low loading, bubbly flows, sedimentation, and cyclone separators. In Eulerian model a set of  $n$  momentum and continuity equations are solved for each phase. The pressure and interphase exchange coefficients, depending on the type of phases involved, are used for coupling. Applications of the Eulerian multiphase model include bubble columns, risers, particle suspension, and fluidized beds.

One of the hydraulic systems which are damaged due to cavitation, are discharge tunnels of dams. An impressive method applied for preventing bubble forming and reducing noise in gated tunnels of dams and also many other hydraulic systems, is injecting another fluid (like air) and so increasing pressure, the process is called aeration [9], [10].

In this paper a two-dimensional cross section of a sample tunnel is modeled in commercial software; ANSYS/FLOTRAN [11]. The modeling is done at different gate openings and the zone of cavitation inception is predicted for each model. Using aeration to reduce cavitation erosion in the tunnel, a vent is considered for aeration after the gate and the models are numerically analyzed again. The RNG  $k - \varepsilon$  model of water-air two-phase flow coupled with the volume of fluid (VOF) method is used in modeling the section both after and before aeration.

R. Jalalabadi is with School of Mechanical Engineering, K. N. Toosi University of Technology (phone: +98912-7390344, email: rjalalabadi@yahoo.com)

N. M. Nouri is with School of Mechanical Engineering, Iran University of Science & Technology (email: mnouri@iust.ac.ir)

It will be shown in results that aeration increases pressure so that the total pressure is more than the amount in which water evaporates and so cavitation is excluded from the system.

## II. THE NUMERICAL APPROACH

Fig. 1 shows two-dimensional cross section of the gated tunnel discussed in this paper. The water flow is modeled at different gate openings, so the gate shown in Fig 1 changes its position and opening becomes greater in seven steps. The gate moves 20cm upward in each step, the flow is numerically simulated and the zone of cavitation inception is predicted.

Seven models are simulated again considering an aerator after the gate. The exact position of the aerator and pressure of aeration is defined using the results which show the place of cavitation inception.

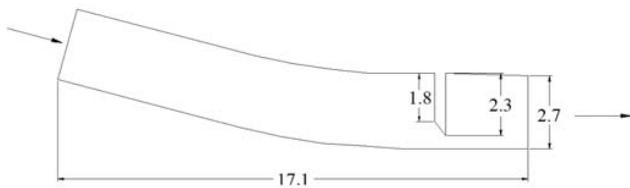


Fig. 1 two-dimensional cross section of the gated tunnel and flow direction, dimensions are in meter

### A. Cavitation Modeling

Two phase cavity flow is modeled by using VOF method here. So it is assumed that both phases have the same velocity in a cell and a series of equations can be used for the flow field like a single-phase flow [12].

In this method a function (f) is used to determine the resulting volume fraction in each cell [1], [13]:

$$f(x,t) = \begin{cases} 1 & \text{in liquid} \\ >0, <1 & \text{at the liquid gas interface} \\ 0 & \text{in gas} \end{cases} \quad (1)$$

The suffix v and l denote vapor and liquid phase and the sum of volume fraction of them equals 1. The transport equation for volume fraction f is [1]:

$$\frac{\partial f}{\partial t} + \vec{v} \cdot \nabla f = \frac{1}{\rho_g} \left( \dot{m}_{lg} - \frac{dp}{dt} \right) \quad (2)$$

The terms on the right hand side of (2) describes the phase change in the system and  $\dot{m}_{lg}$  is the mass transfer rate from liquid to vapor per unit volume.

So density and viscosity is calculated via these formulas [13]:

$$\begin{aligned} \rho &= f\rho_l + (1-f)\rho_v \\ \mu &= f\mu_l + (1-f)\mu_v \end{aligned} \quad (3)$$

Where  $\rho_l$ ,  $\rho_v$ ,  $\mu_l$  and  $\mu_v$  are the density(kg/m<sup>3</sup>) and molecular dynamic viscosity (N·m/s) of vapor and liquid, respectively.

The mass conservation equation for the mixture is:

$$\frac{\partial(\rho_m)}{\partial t} + \nabla(\rho_m \vec{v}_m) = 0 \quad (4)$$

The momentum conservation equation for the mixture is:

$$\frac{\partial(\rho_m \vec{v}_m)}{\partial t} + \nabla(\rho_m \vec{v}_m \vec{v}_m) = -\nabla p + \nabla \cdot [\mu_m (\nabla \vec{v}_m + \nabla \vec{v}_m^T)] \quad (5)$$

The suffix m denotes the mixture. (4) and (5) together with (2) are basic equations used in VOF method.

The algorithm used in ANSYS/FLOTRAN for modeling via VOF method is named VFRC (Volume Fraction). In this algorithm a number is devoted to each cell in the grid. This number which is between 0 and 1 shows the percentage of the cell that is full of main fluid (water); if the number is 0 for a region it means that the region is full of air and if that is 1, the region is full of water. In this software, the governing equations are solved numerically by using control-volume based techniques and SIMPLE algorithm was used for pressure correction.

### B. Turbulence Modeling

Among Reynolds-averaged Navier-Stokes (RANS) equations approach in turbulent modeling, two-equation models are robust and simple in which two separate transport equations for turbulent variables are solved. One of these models is RNG  $k-\varepsilon$  model which is based on the theory of "renormalization group" (RNG) as a mathematical technique used to develop standard k- $\varepsilon$  model [14], [15].

The transport equations for turbulent kinetic energy,  $k$  and its dissipation rate,  $\varepsilon$ , are as follows:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_i k)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + G - \rho \varepsilon \quad (6)$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial(\rho u_i \varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_i} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} G - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \quad (7)$$

Where G, the generation of turbulent kinetic energy due to mean velocity, is calculated using this formula:

$$G = \mu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} \quad (8)$$

In (6), (7), and (8),  $\mu$  is the molecular dynamic viscosity of the fluid (N·s/m<sup>2</sup>);  $k$  is the turbulence kinetic energy (m<sup>2</sup>/s<sup>2</sup>);  $\varepsilon$  is the dissipation rate of  $k$  (m<sup>2</sup>/s<sup>2</sup>). The values of  $c_\mu$ ,  $\sigma_k$ ,  $\sigma_\varepsilon$ ,  $C_{1\varepsilon}$  and  $C_{2\varepsilon}$  as the constants in the above equations are 0.085, 1.42, 1.68, 1.0 and 1.33, respectively.  $\rho$  and  $\mu$  in the above equations are calculated using (3).

### C. Grid Generation and Boundary Conditions

The structured grid generated in the domain is shown in Fig 2 for minimum opening at discharge of the tunnel. For all seven positions of the gate, the grid is similar to Fig 2. The mesh is finer near the walls to study boundary layer better in this region.

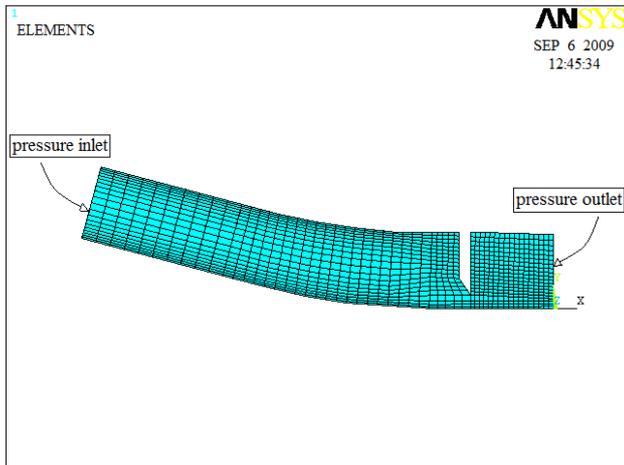


Fig. 2 Grid generated in computational domain and boundary conditions for inlet and outlet

The inlet boundary condition is specified constant pressure which is 1177.2 kPa (gage pressure). Water is discharged to atmosphere so outlet boundary condition is specified pressure too. Ambient temperature is about  $25^{\circ}\text{C}$  and saturated pressure of water in this temperature is -97 kPa (gage pressure). Other boundaries are considered as wall with no slip condition. The boundary value for VFRC is specified as 1 at inlet boundary. It means that all cells in cross section of the tunnel at inlet boundary are filled with water.

In first step, numerical solution is done to predict the zone in which cavitation starts. The pressure and VFRC contours for seven steps of discharge gate will be shown (Figs 4 (a)-(n)). Potential zone for cavitation can be observed by studying pressure and VFRC contours.

In the next step an aerator is placed near the predicted place where cavitation happens. The pressure of air injection and exact point of aerator is determined using pressure contours of first step. Numerical simulation of the geometry for the second time shows that minimum pressure in this condition is greater than the local vapor pressure of water in ambient temperature and so cavitation will not occur (Figs 5 (a)-(n)).

### III. COMPUTATIONAL RESULTS

#### A. Before Aeration

Figs 4 (a)-(n) show pressure and VFRC contours for seven position of discharge gate when there is no aerator in the flow field. The saturated pressure of water at  $25^{\circ}\text{C}$  is -97 kPa (gage pressure).

In Fig 4, minimum pressure in pressure contours is between -140.147kPa to -114.036kPa. So in all cases the pressure is less than -97 kPa and the darkest blue region in the contour is potential zone for cavitation inception. The VFRC contours in the same Fig 4 (a)-(n), show the cells which contain both phases (water and vapor) after the gate.

#### B. After Aeration

In this step, an aerator is placed near the predicted place where cavitation happens, i.e near the gate, so a vent is improvised after and near the gate for entrance of the air. The air is injected to water flow with high pressure through this vent. The pressure of injection and exact point of vent is determined using the data shown in Fig 4.

Aeration pressure should be great enough to increase total pressure in a way that minimum pressure in Fig 4 (-140.147 kPa) exceeds saturated pressure (-97 kPa). So minimum pressure for air injection should be 43.2 kPa, but with this pressure, minimum pressure does not exceeds saturated pressure of water in the flow. So by using trial and error final pressure for air injection is attained 600kPa. The location of the vent is shown in Fig 3. The gate can be larger or smaller, but it would lead to use more powerful equipments to increase pressure and air flow rate due to changing of vent size. Several parameters affect the length of aeration [16] and exact determination of this length should be done using experiments, but here we consider an initial value and finalize it by changing injection pressure simultaneously.

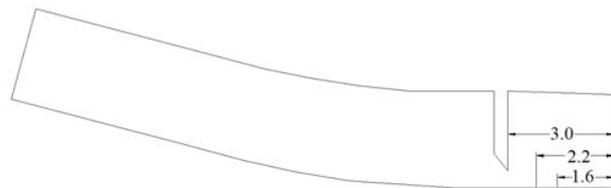
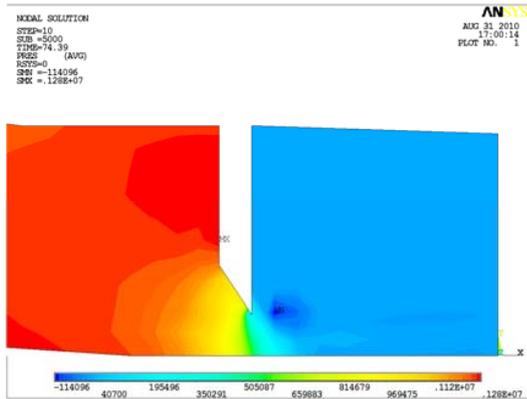
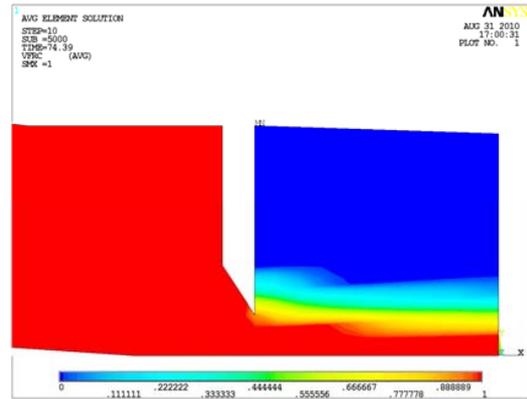


Fig. 3 the location of aerator after the gate

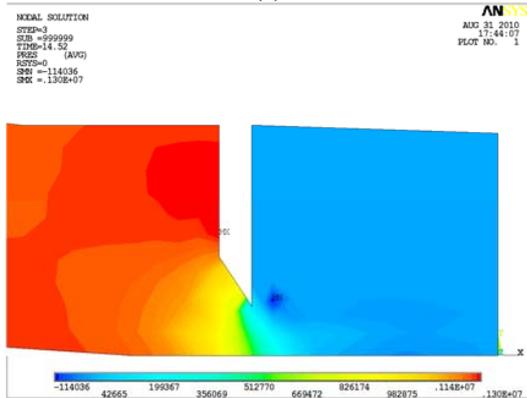
Pressure and VFRC contours for seven steps of discharge gate, considering aerator, are shown in Figs 5 (a)-(n). It can be seen that minimum pressure for all cases is greater than saturated pressure and so cavitation will not occur. VFRC contours also confirm this conclusion.



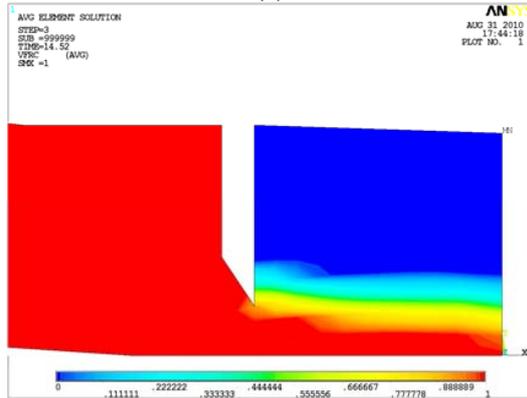
(a)



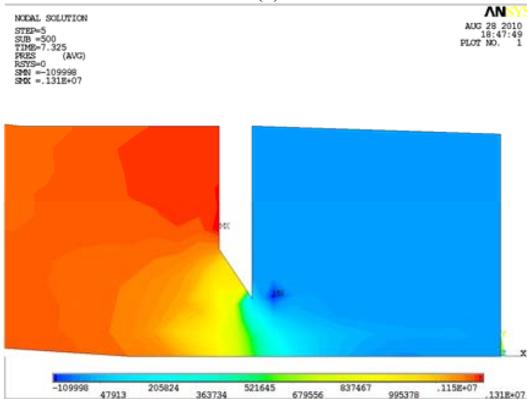
(b)



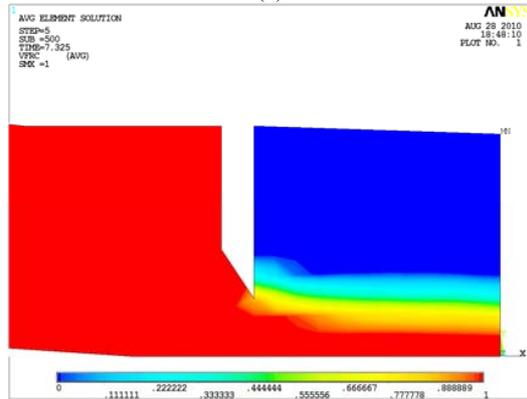
(c)



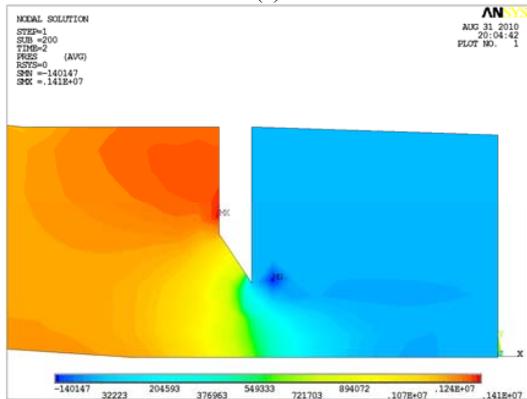
(d)



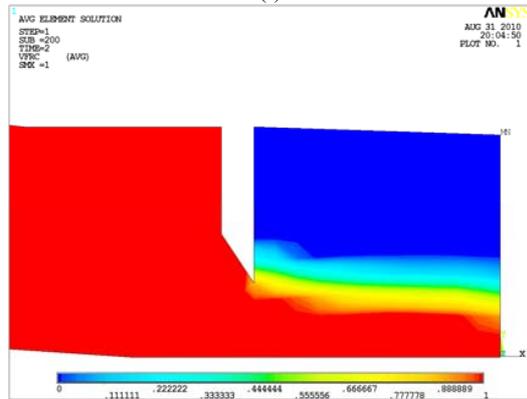
(e)



(f)



(g)



(h)

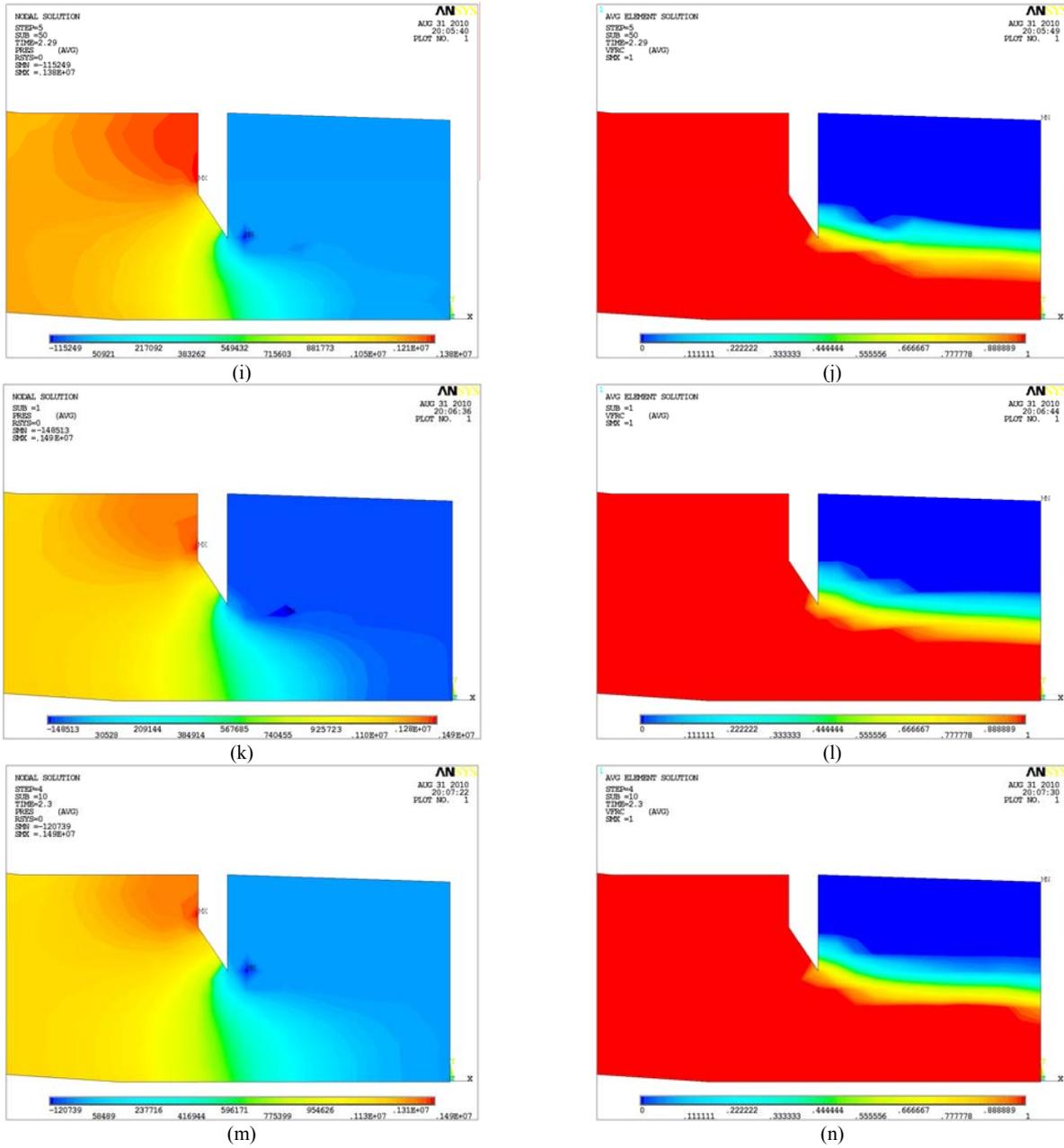
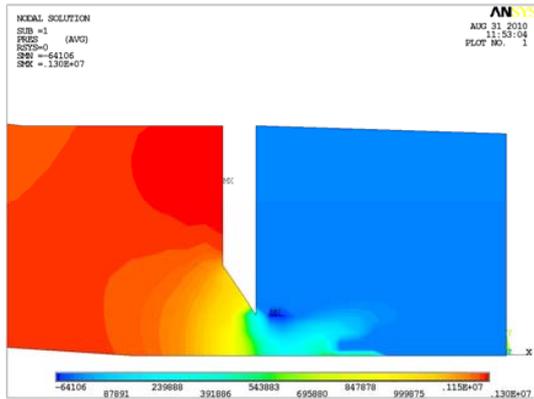
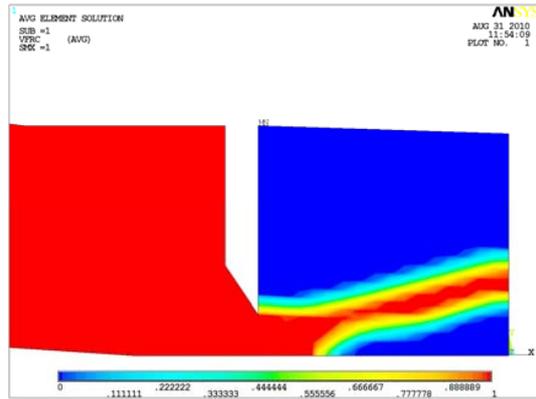


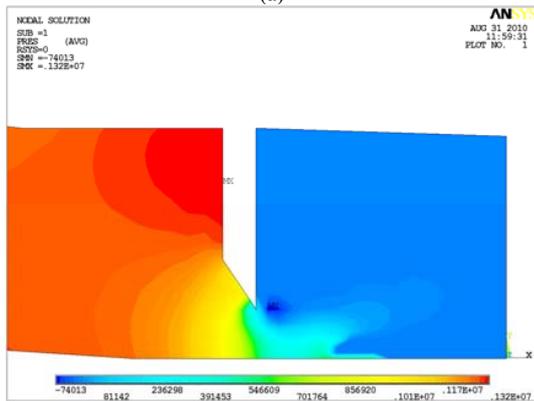
Fig. 4 Pressure and VFRC contours near the discharge gate in seven steps of gate opening without aeration



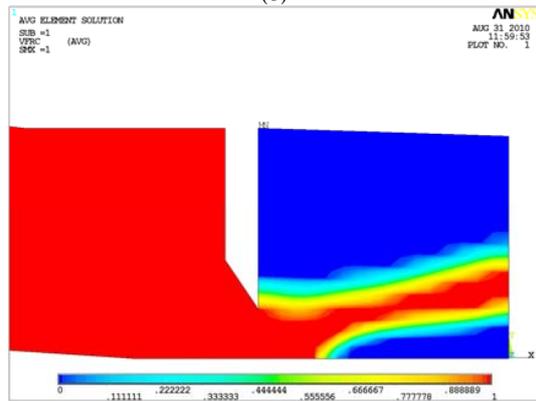
(a)



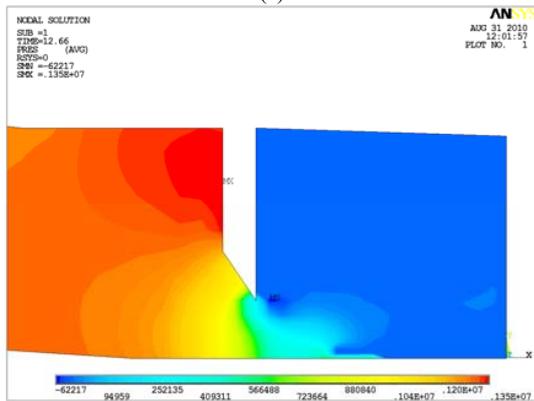
(b)



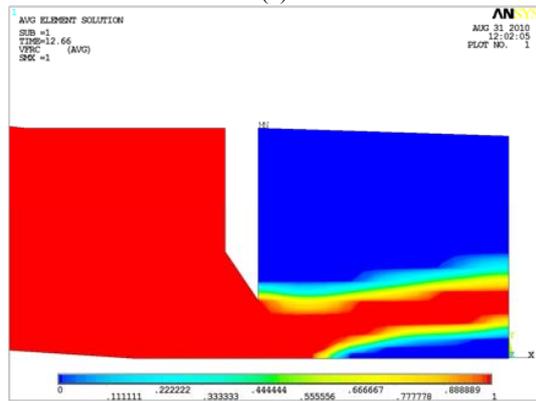
(c)



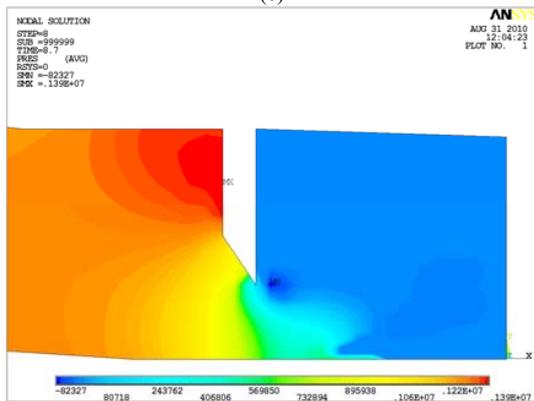
(d)



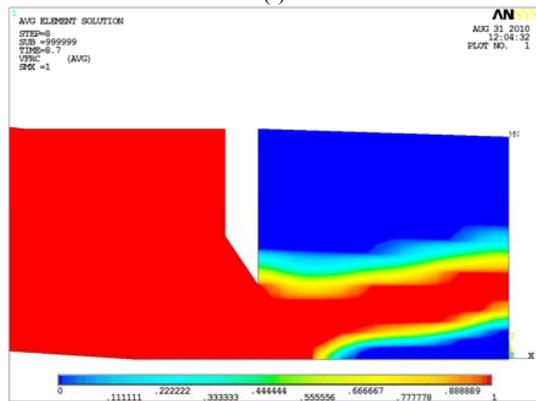
(e)



(f)



(g)



(h)

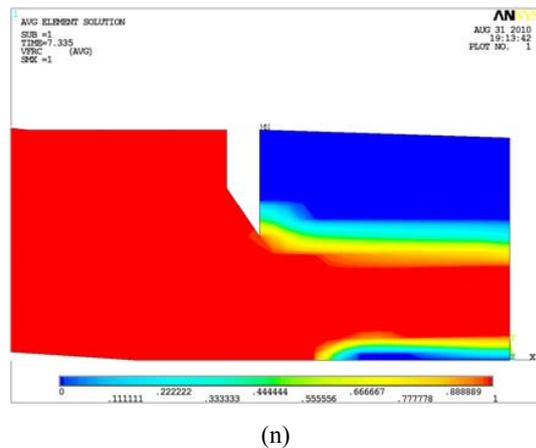
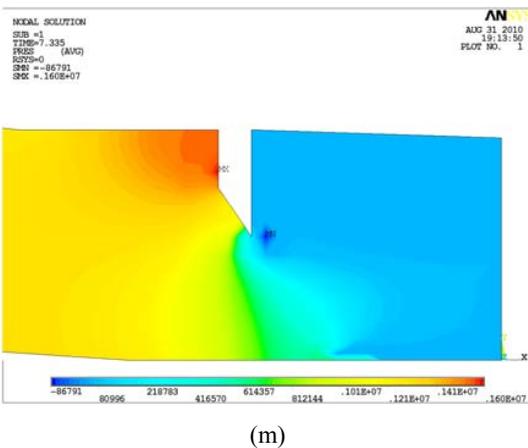
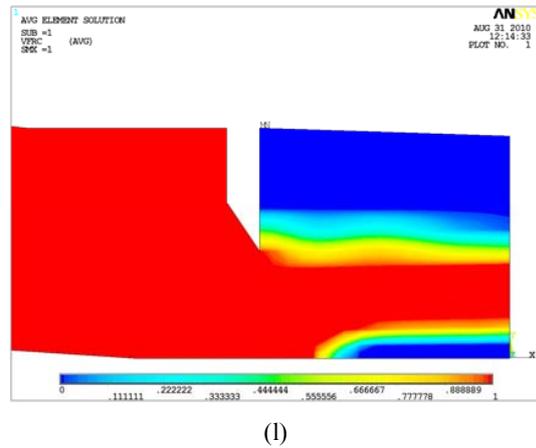
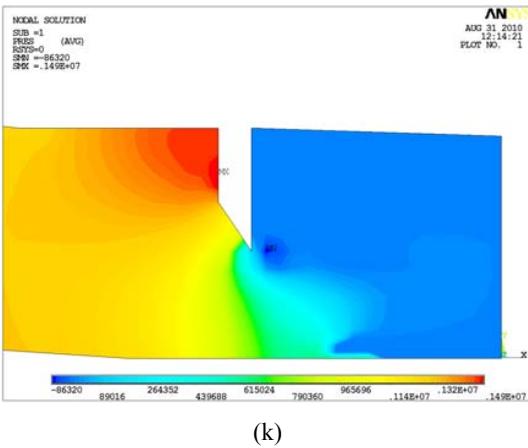
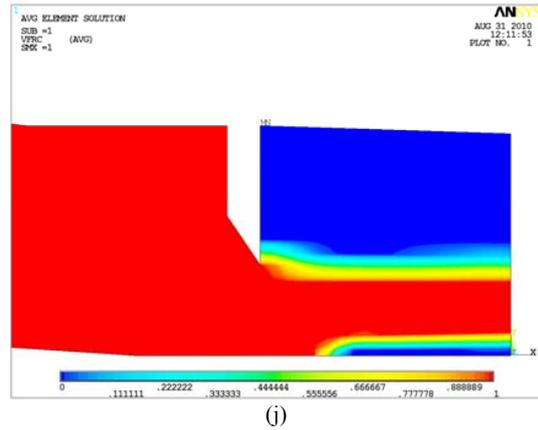
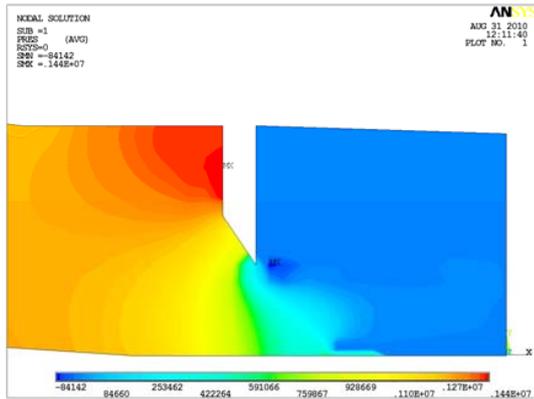


Fig. 5 Pressure and VFRS contours near the discharge gate in seven steps of gate opening with aeration

#### IV. CONCLUSION

Considering the results and pressure contours, it is concluded that aerating is a useful and practical method which can be widely used to exclude cavitation in hydraulic systems like gated tunnels of dams. Numerical simulation can effectively be used to determine the position of cavitation inception and also the position and pressure of aeration with

high precision and costs less in comparison with experimental methods. VOF method, a method that uses transient equations of void fraction with momentum and mass conservation equations, can be effectively useful to study the flow field better. VFRS contours, as results of this method, are helpful both to determine potential zones for cavitation inception in the domain and flow field condition after aeration.

## REFERENCES

- [1] S. Bernard, S. Muntean, R.F. Susan-Resiga, I. Anton, "Multiphase Modeling of Cavitating Flows", *Buletinul Științific AL Universității "POLITEHNICA" Din Timisoara, Transactions on Mechanics*, 2004.
- [2] S. Bernad, R. Susa-Resiga, S. Muntean, I. Anton, "Numerical Analysis Of The Cavitating Flows", In *Proceedings of The Romanian Academy, Series A*, vol. 7, no.1, 2006.
- [3] Y. Xiong, Y. Gao, W. An, "Comparison of Turbulence Models in Predicting Unsteady Supercavitating Flow", *Sixth International Symposium on Cavitation, CAV2006*, Netherlands, 2006.
- [4] F.R. Young, "Cavitation", 1st ed, Imperial College press, 1999.
- [5] FLUENT 6. User's Guide, Fluent Incorporated, 2003.
- [6] C.W. Hirt, B.D. Nichols., "Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries", *Journal of Computational Physics*, vol. 39, pp. 201-225, 1981.
- [7] S.H. Rhee, B.P. Makarov, H. Krishinan, V. Ivanov, "Assessment of the Volume of Fluid Method for Free-Surface Wave Flow", *Journal of Marine Science and Technology*, vol. 10, pp. 173-180, 2005.
- [8] M. R. Najafi, A. R. Zarrati, "Numerical simulation of air-water flow in gated tunnels", *Proceedings of the ICE - Water Management*, vol. 163, Issue 6, pp. 289-295, 2010.
- [9] M. Toorani, "An Investigation of Air Injection effect on Pressure Changes in Super Critical Flows", Bachelor Thesis, K.N. Toosi University of Technology, Civil Engineering Department, 2004.
- [10] G. E. Reisman, M. E. Duttweiler, and C. E. Brennen, "Effects of Air Injection on the Cloud Cavitation of a Hydrofoil", *ASME Fluids Engineering Division Summer Meeting, FEDSM'97*, June, 1997.
- [11] Jahed Motlagh, H.R., Noban, M.R., Eshraghi, M.A., Ansys, 1st Ed, IUST Press, Tehran, pp. 285-297, 2002.
- [12] X. Wang, X. Wang, C. Wang, J. An, Z. Zhang, Q. Duan, "Numerical Simulation of Long-Distance Diversion Tunnel with Free Flow Based on the VOF Method", *Proceedings of International Symposium on Information Science and Engineering*, vol. 1, pp. 19-23, 2008.
- [13] H. Tang, L.C. Wrobel, Z. Fan, "Tracking of Immiscible Interface in Multiple-Material Mixing Process", *Comput. Materials Science*, vol. 29, pp 103-118, 2004.
- [14] V. Yakhot, S.A. Orszag, "Renormalization-Group Analysis of Turbulence", *Journal of Scientific Computing*, vol. 1, no. 1, pp. 3-51, 1986.
- [15] C. Xiangju, C. Yongcan, L. Lin, "Numerical simulation of air-water two-phase flow over stepped spillways", *Science in China Series E: Technological Sciences*, vol. 49, no. 6, pp. 674-684, 2006.
- [16] A., Mohaghegh, J. H. WU, "Effects of hydraulic and geometric parameters on downstream cavity length of discharge tunnel service gate", *Journal of Hydrodynamics, Ser. B*, vol. 21, Issue 6, pp. 774-778, 2009.