Investigating the Effect of The Ramp Geometry on the Hydraulic Characteristics of The Flow Around the Ramp Embedded in the Under Pressured Duct

Hamzeh Ebrahimnezhadian[1], Mohammad Manafpour[2]
1- PhD of Water and Hydraulic Structures, Civil Engineering Department, Faculty of Engineering, Urmia University, Urmia, Iran
2- Assistance Professor of Hydraulic Structures, Civil Engineering Department, Faculty of Engineering, Urmia University, Urmia, Iran

Email: h.ebrahimnezhadian@urmia.ac.ir

Abstract

The flow downstream of the ramp embedded in the bottom of an under pressured duct is divided to several zones including the recirculation, shear layer and the main flow zones. The angle and height of ramp as the geometrical components can affect the flow characteristics including the length of the recirculation zone, flow velocity and pressure coefficients. In the present study, the hydraulic characteristics of flow over the ramp mounted on the bed of an under pressured duct are numerically investigated where the range of angle and ramp height variation are considered respectively $5^\circ \geq \theta \geq 20^\circ$ and $0.1 \leq t_r/d \leq 0.4$. The flow field is numerically simulated using OpenFOAM open source software, MultiPhaseEulerFoam solver and various turbulence models. In order to validate the performance of the numerical model, the results of Manafpour (2004) tests are applied. Comparison of numerical and experimental results shows that the least difference is found for K-ω SST two-equation turbulence model in which the maximum and minimum difference of the results are equal to 7.14% and 0.07% in the ramps C and A respectively. By increasing the height and angle of the ramp, both the recirculation length and the maximum turbulence intensity of the flow are increased while the minimum pressure level of the recirculation zone is reduced. However, the sensitivity of all these parameters to the ramp height is high in comparison to the ramp angle.

Keywords: Ramp geometry, under pressure duct, OpenFOAM, K-ω SST turbulence model

1. INTRODUCTION

The flow over a ramp is a prototype for separating, recirculating and reattaching flow in nature and numerous engineering applications. Examples include the flows around buildings, inside combustors, industrial ducts and in the cooling of electronic devices. All of those cases share one common feature: That an adverse pressure gradient (usually due to a sudden change of geometry) causes the boundary layer to separate from the surface and form a mixing layer, which eventually re-attaches to the surface. The presence of separation, recirculation and reattachment drastically changes the transport of momentum and heat. The flow around ramp in under pressure duct similar to backward-facing step is a prototype of these scenarios, as it demonstrates the phenomena with a simple geometry, one that is easy to set-up experimentally, as well as model computationally.

In under pressure flow upon ramp aerator embedded in the tunnel floor, immediately downstream of the ramp such as Fig. 1, the flow is two-phase, disturbed with no surface aeration and divided into the various zones immediately downstream of the ramp, including the cavity zone and the main zone of flow above the shear layer [1],[2].

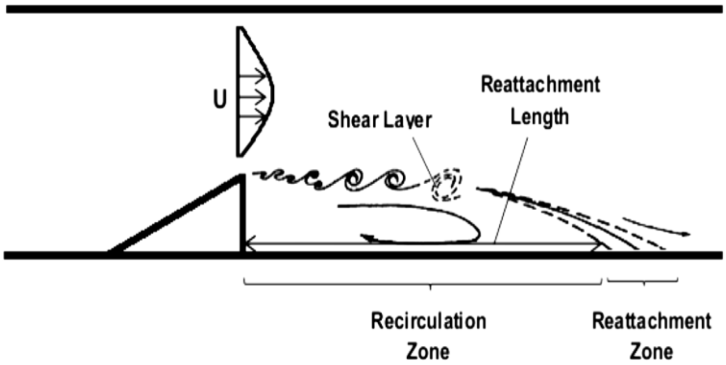

Figure 1. Flow pattern upon under Pressure tunnel ramp

As flow separates from the trailing edge of the ramp, the lower layers accelerate and small Intense eddies form on the lower surface of the nappe which roughens the air-water interface, Fig. 1. As turbulence overcomes surface tension, this air will be entrained into the flow. On the other hand, the pressure distribution in the nappe rapidly adjusts itself to near atmospheric conditions and the slip velocity of air bubbles falls and air bubbles are diffused by turbulence deeper into the core of the nappe. Further downstream, more and more air will be entrained into the flow [1]. Turbulence plays an important rule here, and higher turbulence levels cause more air entrainment [2] ,[3],[4], [5].

Proper recognition of the two-phase flow pattern in under pressure tunnel spillways with and without aeration system is among the topics studied by a number of researchers, and because of the complexity of two-phase flows, there is still a need for more laboratory and numerically studies in this field. Part of the studies on the flow around the ramp aerator in pressure tunnels which are often also provided in laboratory form is expressed in next section.

Zheng et. al studied and compared VOF and Mixture methods during a series of experimental and numerical studies with three-dimensional simulation of aerated flow downstream of ramp aerator in a under pressure tunnel. Comparing the numerical and experimental results indicates that both methods accurately calculate the free water surface and cavity length. Air concentration inside the water and the average values of pressure downstream of aerator from Mixture model have less error than the experimental results [9].

Kavianpour et. al carried out laboratory studies on measuring the pressure fluctuations in downstream of the aerator located in a circular tunnel and studied aerator performance with different geometries. According to the results of the experiments, it was found that the use of aerator with small heights for air supply in tunnel duct is more suitable than aerator with high heights from a hydrodynamic point of view [6].

Narayanan et. al carried out Experiments to study the effect of air injection on the wall pressure in the near field of deflectors placed in a duct. Various quantities of air have been injected just downstream of the ramps to form stationary air cavities. The variation of cavity length has been studied as function of the ratio of the volume of air flow to the water discharge. The wall pressure field in terms of the mean and fluctuating parts is presented. This study shows how air injection affects the wall pressure field from that without air injection. Information concerning wall pressure is expected to be useful in the design of deflectors in tunnels and spillways [7].

In this study, in order to investigate the effect of dimensionless geometry parameters on flow characterestics, the hydraulic characteristics of flow over the ramp mounted on the bed of an under pressured duct are numerically investigated where the range of angle and ramp height variation are considered respectively $5^{\circ} \geq \theta \geq 20^{\circ}$ and $0.1 \leq t_r/d \leq 0.4$.

2. MATERIAL AND METHODS

Experimental setup: The details of experimental arrangement have been described in Fig. 2. The experimental model consist of a horizontal pressurized duct with a square cross section that ramps of Perspex of triangular cross section were glued to the bottom of duct. Four ramps were used for the study. Two heights of t_r/d=0.1 and 0.2 were used. tr is the height of the ramp and d is the height of the duct. For each value of t_r/d test, two slopes of 5^o and 10^o were used. Water supply to the duct housing the ramps was from a constant head tank located on the roof of the laboratory. A 0.1m wide and height duct was used with a tank upstream to achieve the desired flow velocities. The modeled aerator consisted of a ramp. Dimensions of the aerator are given in Table 1. The aerator was installed 1m downstream of the channel entrance to allow the turbulent boundary layer to develop completely. To measure flow velocity a Pitot tube was employed. Air was introduced through nine holes of diameter 10mm placed at regular intervals immediately downstream of the ramps [8].

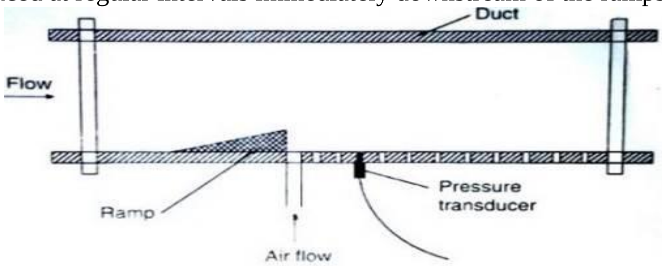

Figure 2. Experimental apparatus

Table 1- Characteristic of Studied Ramps

Ramp	Ramp height t_r(mm)	Ramp Length L_r(mm)	Ramp Angle(θ)	t_r/d
A	10	113.4	5	0.1
B	10	56.7	10	0.1
C	20	226.8	5	0.2
D	20	113.4	10	0.2

Numerical Model: In the present study, OpenFOAM open source software was used to simulate the flow around the aerator ramp embedded in a under pressure duct.

OpenFOAM: The widely known CFD-toolbox “OpenFOAM” (Open Field The official version of OpenFOAM is distributed under the GNU license by ESI/OpenCFD (www.openfoam.org). For post-processing results of OpenFOAM simulations, the open source software ParaView can be used. With ParaView, even domain decomposed cases (that were calculated in parallel on several CPUs and are stored in separate directories for each domain), can be post-processed without reconstructing the case. In contrary to most CFD programmes, OpenFOAM is not delivered with a graphical user interface for performing the pre- and post-processing of the simulations [9],[10].

The most important difference between open source software OpenFOAM and other computational fluid dynamics commercial software, including Fluent and CFX, That is, in OpenFOAM software to create a model in each subcategory, the indices needed to carry out the software project must first be defined and then use an exclusive solver, while in other software mentioned, only the indices needed to create the model are included in the software and immediately the problem is solved. Therefore, one of the difficulties of open source software OpenFOAM is the choice of proportional solver. The flow around the under pressure tunnel ramp aerator have two-phase nature. Multiphase flow modelling is performed using the three views, 1- Volume of Fluid 2- Eulerian-Lagrangian 3- Eulerian- Eulerian, which the VOF method is used to solve multiphase systems involving two or more immiscible fluids with emphasis on capturing the interface and can use for problem such as dam break and Jet breakup, in the Eulerian-Lagrangian method, continuous-phase from the Eulerian Approach and the dispersed phase (motion and particle tracing) from the Lagrangian Approach is modelled. This method is suitable for two-phase flows with less than 10% Volumetric of dispersed phase and in the Eulerian-Eulerian Approaches, both continuous and dispersed phases are considered as continuous phases, this method is used for industrial systems with billions of solid particles, with a dispersed phase content of more than 10% [11].

In OpenFOAM software, to analyze multiphase flow problems, various solvers are foreseen. Choose a suitable solver is the most important part of the simulation in OpenFOAM open source software. Table 2 shows a number of multiphase solvers in the OpenFOAM.

Table 2- Multiphase Solver in OpenFOAM

Multiphase solvers	Description
interFoam	solver for two incompressible fluids capturing the interface using a VOF method
bubbleFoam	solver for 2 incompressible fluids to compute dispersed gas-liquid and liquid-liquid flows based on the Euler-Euler two-fluid methodology
twoPhaseEulerFoam	solver for 2 incompressible fluid phases with one phase dispersed based on the Euler-Euler two-fluid methodology

In this study, due to the two-phase nature of the flow and the importance of mixing the two phases in each other, the Eulerian- Eulerian approach and also a solver that have the basis of the twoPhaseEulerFoam solver is used and the necessary development in the solver to increase efficiency in the discussion of the two-phase mixing simulations has been done.

Geometry, Meshing and Boundary Condition: In order to construct the geometry of the numerical model, due to the various configurations in the present study, including the angles and height of the ramps, a code was written in the FORTRAN software. By applying the input data such as the ramp angle and height, the distance to the ramp entrance and the distance from the end of the ramp, the information necessary for constructing geometry from the FORTRAN code was extracted and transferred to the BlockMesh environment inside the OpenFOAM software. Regarding the dimensions of the study area and air bubble diameter, the geometry grid of the numerical model with dimensions of 2 mm was prepared. The meshing of the numerical model is visible in Fig. 3. The sensitivity analyze of the numerical model was measured relative to the mesh size, and it was found that using a smaller cell size did not affect the accuracy of the calculations.

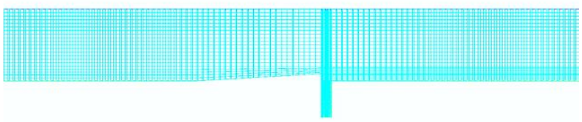

Figure 3. Numerical Model Meshing

The boundary conditions in the present study include the following; 1-water inlet: mass flow rate 2-outlet: pressure outlet and for all duct wall such as floor, ceilings and ramp, a wall boundary condition were used.

3. RESULT AND DISCUSSION

In order to evaluate the performance of a numerical model in the simulation of two-phase flow around aerator ramp embedded in under pressure duct bed for 4 types of ramps that are presented in Table 1, numerical and experimental results for number of parameters such as the Recirculation length immediately downstream ramp, the average flow velocity profiles and the changes in the pressure coefficients (C_P) were compared. In the next step, the flow near the ramp embedded in a pressurized flow under non-aeration conditions for a change in the ramp angle in 4 scenarios (5, 10, 15, 20) degrees and ramp height in 4 scenarios (0.1, 0.2, 0.3, 0.4) m were simulated, in order to determine the effect of the ramp angle and height on the flow characteristics such as the recirculation length and the minimum values of the floor pressure immediately below the ramp.

4. VERIFICATION AND NUMERICAL RESULTS

The Recirculation Length (L_r): After the passing flow through the aerator ramp, in addition to the main zone, the recirculation zone is formed that this two regions are separated from each other by the shear layer between them as shown in Fig. 4. In Figure 5, the comparison between the relative recirculation lengths calculated by the numerical model Using the two-equation turbulence model K-ω SST and experimental results for cavity formed immediately downstream of four different ramps is showed in which L_r and t_r are the length of the recirculation zone and ramp height Respectively. According to Figure 5, the two-dimensional numerical model in all ramps estimates the cavity length less than the experimental model. The correlation coefficient between the numerical and laboratory model results is 0.90. The low range of discrepancies obtained for the recirculation length indicates the acceptable simulation of the flow by the numerical model.

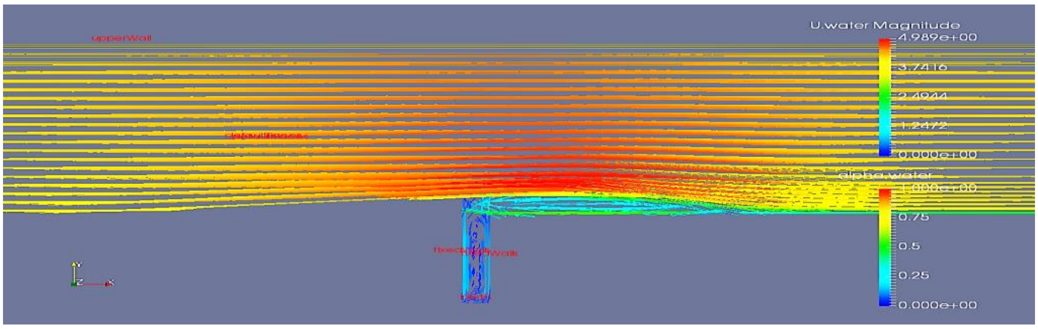

Figure 4. Cavity formed in the under pressure duct due to forced air injection.

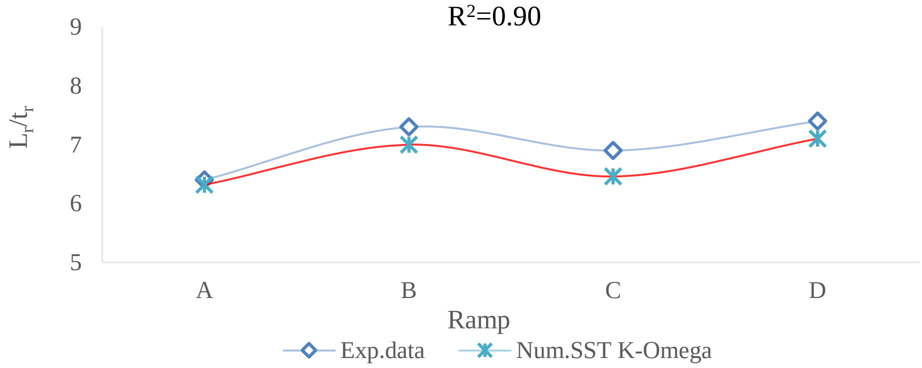

Figure 5. Comparison of the relative recirculation lengths of Numerical and Laboratory Model Results and Correlation between Results.

Flow Velocity: The comparison of average velocity distribution profiles between numerical and experimental results for ramp D ($t_r/d=0.2$, $\theta=10^o$) is displayed in Figure 6 where Z is the distance from the end of the ramp and U is the average velocity of the flow that equal to 4.1 m/s. The adaptation of numerical and experimental results suggests the proper accuracy of the numerical model in simulating the flow velocity field near the aerator ramp. According to Figure 6, the velocity in the direction of flow at the end of the ramp due to the Existence of ramp and the decrease of the duct cross section reaches its maximum value.

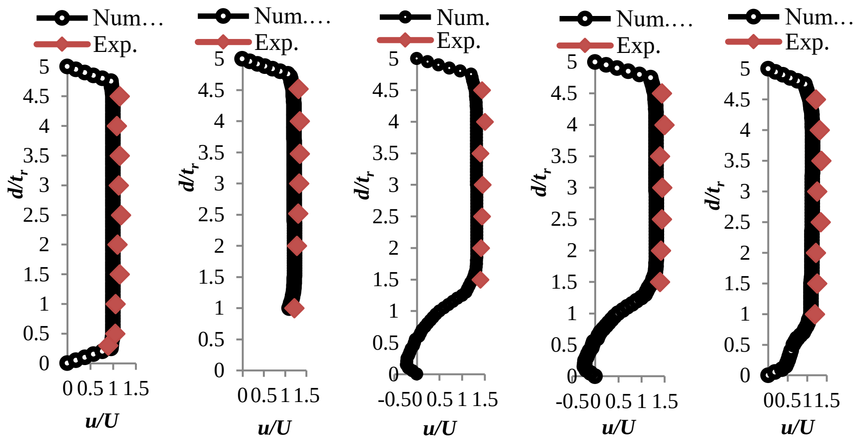

Figure 6. Flow velocity Profiles for ramp D ($t_r/d=0.2$, $\theta=10^o$), in section (Z/t_r: a=-28, b=-0.2, c=2.1, d=6.7, e=15.7)

Pressure Coefficient: The dimensionless parameter of the pressure coefficients (C_P) is defined as the static pressure ratio to the dynamic pressure. In this research, the pressure coefficient Obtained from the relationship $C_P=P-P_o/0.5\rho U^2$ where in, P is the Static pressure of the floor at the desired point, P_o is the Static pressure at the reference point located at the upstream of ramp, ρ is the water density and U is the average flow velocity. Figure 7 shows the variation of the pressure coefficient in the duct floor relative to the distance from the downstream of ramp. The correlation coefficient between numerical and experimental results for pressure coefficient of the ramp B ($t_r/d=0.1$, $\theta=10^o$) is 0.9167 which emphasizes the accuracy of the numerical model in simulating the flow field.

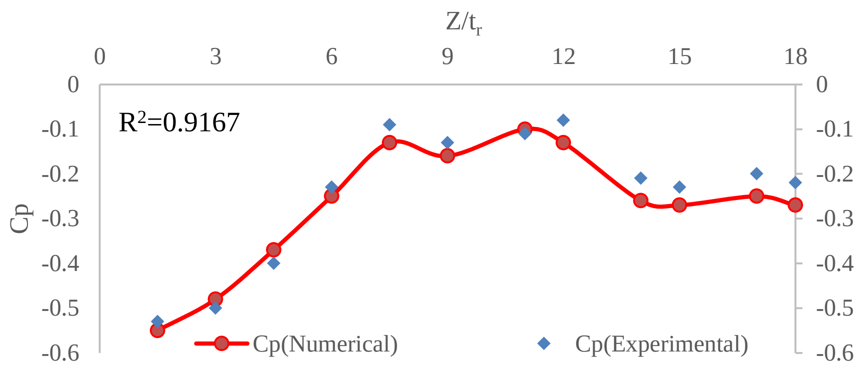

Figure 7. Changes in the pressure coefficient in the duct floor relative to the distance from the end of the ramp B (t_r/d=0.1, θ=10º)

5. EFFECT OF ANGLE AND HEIGHT OF RAMP ON RECIRCULATION LENGTH

To investigate the effect of angle and ramp height on flow characteristics, change in ramp angle in 4 scenarios (5, 10, 15, 20) degree in constant relative ramp height t_r/d=0.1 and ramp height in 4 scenarios (0.1, 0.2, 3 0.4 m in constant ramp angle of θ=5º were used as the criterion. Figures (8a and b) show the effect of increasing the ramp height in fixed ramp angle θ=5º and increasing the angle of the ramp in fixed relative ramp height t_r/d=0.1, respectively. The results show that the values of the recirculation length increase in both cases the constant height of the ramp and its increasing angle and the constant angle of the ramp and increasing its height. However, with increasing height at a constant angle of the ramp, the recirculation length increases with higher intensity, that indicates a greater sensitivity of the recirculation length to the height of the ramp.

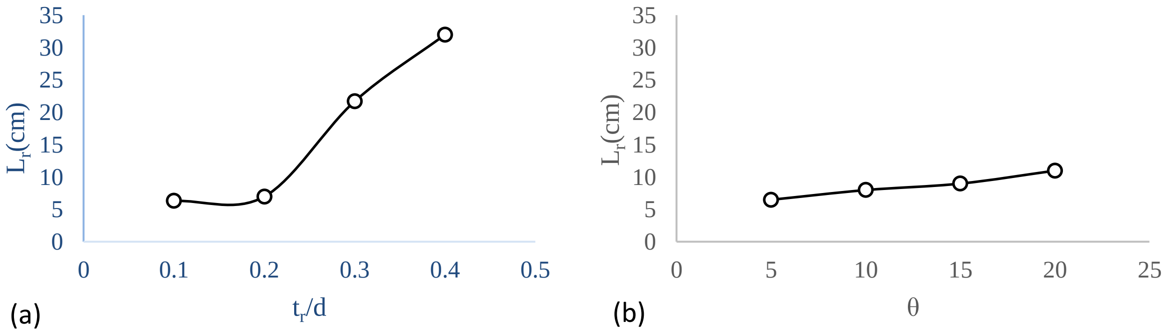

Figure 8. Changes in recirculation length: (a) Ramp height increase relative to fixed ramp angle 5º , b) ramp angle increase relative to fixed ramp height (t_r/d=0.1).

6. EFFECT OF ANGLE AND HEIGHT OF RAMP ON MINIMUM PRESSURE COEFFICIENT

The effect of increasing relative height and ramp angle on C_{Pmin} follows [17], which showed that with increasing ramp height and angle, the local pressure in the whole duct, decreases. The effect of increasing relative height and ramp angle on C_{Pmin} at the recirculation zone created at the downstream of the ramp is shown in Figure 9. C_{Pmin} values in both cases are reduced and the impact intensity of the C_{Pmin} of the ramp height (t_r /d) is far greater than ramp angle. Therefore, the sensitivity of the minimum pressure coefficient to increasing ramp height is higher than the ramp angle's.

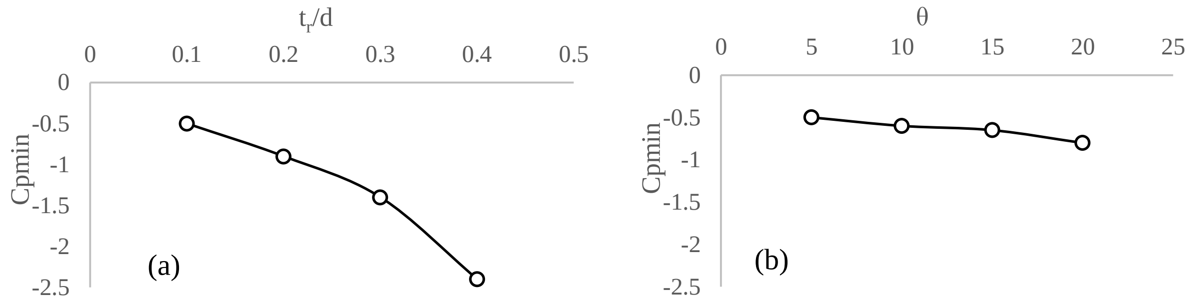

Figure 9. Minimum floor pressure of cavity zone variations: a) Increasing ramp height for fixed ramp angle 5º , b) increasing ramp angle for fixed ramps height (t_r/d=0.1).

7. CONCLUSIONS

This study on the flow field near the aerator ramp of a under pressure duct without aeration by OpenFOAM produced the following results:

The correlation between numerical and experimental results for recirculation length and pressure coefficient is 0.90 and 0.91 respectively, which indicates proper agreement between numerical and experimental results and the superiority of the TwoPhaseEulerFoam solver in the simulation of two-phase flow.

In both cases of fixed ramp height and increasing ramp angle and fixed ramp angle and increasing ramp height, the values of the cavity length and maximum turbulence intensity increase, and the minimum pressure values at the recirculation zone bed are reduced. However, with increasing ramp height at fixed ramp angle, the intensity of the increase in the recirculation length, and the reduction of the minimum pressure at the recirculation zone bed are greater indicating higher sensitivity to ramp height.

8. REFRENCES

1. Attari, J., and Zarrati, A.R., (1997). The effect of nappe impact angle on aerator performance. 27th IAHR Congress, San Francisco, USA.

2. Chanson, H., (1989). Flow downstream of an aerator. Aerator spacing. Journal of Hydraulic Research, IAHR, Vol. 27, No. 4, pp. 519-536.

3. Ervine, D.A., and Khan, A.R., (1991). Turbulence measurements in an air-slot ramp aerator. 24th Congress of IAHR, Madrid, Spain, Vol. C, pp.73-79.

4. Falvey, H.T., (1990). Cavitation in chutes and spillways. USBR Engineering Monograph, No. 42, Denver, Colorado, USA, 160 pages.

5. Peterka, A.J., (1953). The effect of entrained air on cavitation pitting. Joint Meeting Paper, IAHR/ASCE, Minneapolis, Minnesota, Aug., pp. 507-518.

6. Kavianpour, M., Toloui, A., and al-Mu'zsoumi, A.S., (2002). Evaluation of pressure fluctuations in downstream of aerator in lower tunnels of dams with circular sections. National Conference on National Hydroelectric Power Plants, June 6 and 7, Tehran (in Persian).

7. R. Narayanan, M. Manafpour, M. R. Kavianpour, (2001). Wall pressure field in the reattaching flow past deflectors, part 2: with air injection", in: Proceedings of the Institution of Civil Engineers-Water and Maritime Engineering, Vol. 148. No. 1, pp. 39-45, Thomas Telford Ltd, 2001

8. M. Manafpour, (2004). Effects of Air Injection on the Flow Immediately Downstream of Deflectors on Spillways, PhD Thesis, Manchester University, UK.

9. Zhang, JM, Chen JG, Xu WL., (2011). Three-dimensional numerical simulation of aerated flows downstream sudden fall aerator expansion in a tunnel. Journal of Hydrodynamics. 23(1): 71–80.

10. Zhang, X. D., (2004). Three dimensional numerical simulation of high velocity flow in spillway tunnels, [Ph.D. thesis] , China Institute of Water Resources and Hydropower, Beijing, China.

11. Weller, H. G., Jasak, H., and Fureby, C., (1998). A tensorial approach to computational continuum mechanics using object-oriented techniques. Journal of Computational Physics, 12 (6), pp. 620–631.