Study of Two-Phase Flow Behavior around Chute Aerator Using OpenFOAM
Hamzeh Ebrahimnezhadian*, Mohammad Manafpour*

*Department of Civil Engineering, Urmia University, 11Km Sero Road, Urmia, 5756151818, Iran
Corresponding author: *h.ebrahimnezhadian@urmia.ac.ir

Abstract—Aeration devices are installed on chute spillways to prevent cavitation damage in high-velocity flows. Large quantities of air-entrained characterize bottom aerators along with the jet interfaces and a strong de-aeration process near the impact of the water jet with the spillway bottom. Appropriate prediction of the air entrainment process, flow characteristics, and two-phase flow pattern at the aerator would contribute to reliable spillway operation. The mathematical formulation of two-phase flow at an aerator remains a challenging issue for spillway design due to its complexities. In the present study, 2D numerical simulations are performed to predict the distribution of air concentrations, flow velocity, turbulent intensity, and water surface profile along the chute aerator using open-source OpenFOAM software and RNG k-ε turbulent model. The correlation coefficients obtained between the numerical and experimental results indicate that a proper agreement exists between the relative cavity length and velocity profile results. The research results show that the turbulent intensity of flow passing over the aerator ramp is significantly increases. It means that the ramp acts as a turbulent generator on the spillway chute. The maximum value of turbulence intensity occurs at the 6-8% of flow depth(h) from the tip of the ramp, and it is raised as the Froude number increases. It is concluded that the intake air flow rate and air-entrainment coefficient (β) are linearly increased concerning the Froude number.

Keywords— Aerator ramp; air entrainment; cavity length; RNG k-ε turbulence model; OpenFOAM.

I. INTRODUCTION
Spillways and chutes operating under high-velocity flows are at risk of the cavitation phenomenon, which causes major damages or endangers the dam stability. Therefore, in engineering designs, it is necessary to protecting spillways against cavitation damage [1]. Control of cavity bubbles collapse near the solid boundary and use of cavitation-resistant materials are some approaches for preventing cavitation damage [2]. Aeration is the most efficient and economical method for preventing cavitation in high-speed flows over chute spillways [3], [4]. With 8% air near the concrete surface, damage of cavitation attack is completely prevented [5]. Aeration can occur in both natural and induced forms. Air entrained through flow-free surface results in a “white water” region at the free surface and is not sufficient to reduce the damage risk of cavitation phenomenon on solid boundary [2]. Free-surface aeration occurs whenever the thickness of the turbulent boundary layer reaches the depth of flow. Free-surface aeration alone is not sufficient to avoid or reduce the possibility of cavitation damage [6]. Injecting air into the flow is one of the best and most economical ways to protect the surface of the spillway against cavitation damage. Therefore, with regard to the danger of cavitation attack, induced aeration of flow is recommended [7]. The flow is separated from the spillway surface by a geometrical device, which forms a nappe and leads to aerated flow, as shown in Figure 1. These devices, which are used to inject air into the flow, are called aerators and are usually installed on the spillway floor and also on the side walls (Figure 2) [2], [8].
A small deflection on a chute bed (e.g., ramp) tends to separate the flow away from the chute surface. In the cavity formed below the nappe, a local sub pressure (∆P) is produced by which air is dragged into the flow (Q_{air} inlet). The main flow regions in the vicinity of a bottom aerator are the approach flow region, the transition region, the aeration region, reattaching region, and the downstream flow region (Figure 3) [9].

Researchers in the field of hydraulics have studied aeration tools such as ramps for years, both in the laboratory and field observations [11]. Since aerators’ field observations are associated with limitations, laboratory studies have been used by researchers as a major research tool in this field for many years. Despite the prevalence of laboratory research, its cost and timeliness have always been a major problem. Researchers have done a lot of research to achieve empirical equations to estimate the characteristics of aerated flow. However, the equations are generally not practical in design [12], [13]. Kramer and Hager [14] investigated the flow characteristics such as velocity, air demand, and air bubble size distributions experimentally. They found that the bubble rise velocity in chute flows on the Froude number. Computational fluid dynamics is used as an essential complement to laboratory studies in two-phase flow modeling. CFD helps us to investigate the aerated flow field numerically in detail. Some researchers who employed the VOF and Mixture models to study an aerated flow found that the Mixture model is more suitable for simulating this type of flow, especially with high air concentration [15]. Zhang [16] carried out three-dimensional modeling using the Mixture approach. He evaluated aerated flow characteristics such as air bubble diameter; he found that smaller air bubble diameter leads to better agreement with experimental results. Ruidi et al. [17] examined the air concentration and bubble characteristics downstream of the chute aerator laboratory. Based on their research results, they presented a formula for the maximum frequency of air bubbles. Teng et al. [18] created a 3D numerical model using the VOF approach for flow over MOFORSEN dam aerator and compared the results with field measurement. The results show good agreement between the flow field measurement and numerical models. Lian et al. [19] investigated air entrainment and air demand in the Spillway Tunnel using numerical models, laboratory experiments, and field measurement. They found a remarkable difference between the results from the prototype and laboratory experiments, which indicates the scale effect in the physical model. Yang et al. [20]-[22] conducted numerical research and used fluent software to investigate the effect of flow aeration in wide spillway aerators. The results showed that wide spillway aerators play a significant role in flow aeration, and this effect has a higher percentage difference near the wall and chute center zones than normal spillway aerators. By conducting a series of experiments, Ruidi et al. [23] examined the characteristics of the flow around the aerator, including the amount of air entering the flow. The results showed that air entrainment is directly related to the Froude number. Sarvar et al. [24] evaluated orifice spillway aerators’ function with different slopes using a numerical and experimental model. They developed an empirical equation based on results that will be appropriate for the orifice spillway aerators with a slope from 26° to 32°. Yang et al. [25] studied two-phase flows in a large chute aerator numerically. The result showed that for characteristics such as cavity length and air ratio, the better adaption of laboratory and numerical results occurs when smaller air bubble diameters are used. Cihan Aydin [26], [27] performed various laboratory tests for different Froude number and ramp heights to evaluate spillway aerators’ performance. This study’s results led to experimental relationships for estimating the air intake coefficient, which showed high compliance with laboratory results.

This study aims to investigate numerically the characteristics of flow passing an aerator ramp installed on the chute bed. Special attention are given to determine the distribution of flow velocity, air concentration, and turbulent intensity along the chute. Also, the effect of the flow Froude number.
number on turbulent intensity (Tu), air discharge (Qa) and aeration coefficient (β) is examined.

II. MATERIALS AND METHOD

A. Experimental Set-up

The experimental information was gained from a hydraulic model of an aerator made in a flume of 0.20 m wide at the Laboratory of the School of Engineering, São Carlos, Brazil (Figure 4) with the following specifications [28].

| Runs | Opening of the floodgate (cm) | Water flow rate, Qw (l/s) | Froude Number | Cavity Length (m) | Air Flow rate, Qair (l/s) |
|------|-------------------------------|--------------------------|---------------|------------------|----------------------------|
| 3    | 6                             | 47.65                    | 6.31          | 1.08             | 12.66                      |
| 5    | 6                             | 64.37                    | 8.19          | 1.48             | 20.17                      |
| 8    | 9                             | 64.38                    | 5.50          | 0.98             | 28                          |
| 9    | 9                             | 98.20                    | 6.55          | 1.48             | 21.4                        |
| 12   | 11                            | 64.38                    | 4.59          | 0.88             | 25.8                        |

B. Numerical Model

Computational Fluid Dynamics (CFD) is an Engineering tool that uses numerical methods to model and analyze physical phenomena that dealt mostly with high-velocity fluid flow and complex geometry. In recent decades application of computational fluid dynamics has grown substantially. The use of Computational Fluid Dynamics, which results in time and cost-saving, eliminates the experimental limitations and produces supplementary data to those of laboratory ones.

Open-source software allows users to access the application code and provide a cheap approach for simulations, compared to the commercial software. However, open-source software is dependent on more skilled users in comparison to commercial software. In the present study, OpenFOAM open source software is used to simulate the 2-phase flow around a ramp aerator mounted on the floor of a steep channel.

C. OpenFOAM

The widely known CFD-toolbox “OpenFOAM” is a free and open-source CFD software distributed under the GNU license by ESI/OpenCFD [29]. OpenFOAM development and the number of users has increased significantly due to issues as follows:

- a well-designed C++ library that allows the numerical simulation of various Engineering problems.
- Its object-orientated structure is very flexible and can be adjusted to very specific problems. Since the code is open-source, code analysis and manipulation are possible, and to discretize and solve complex fluid problems, it uses Finite Volume Methodology [30].

One of the difficulties of open source OpenFOAM software is the choice of the proportional solver. The free surface flow over ramp aerator has two-phase nature. Multiphase flow modeling is performed using the three views, 1- Volume of Fluid 2- Eulerian-Lagrangian 3- Eulerian-Eulerian. In this study, due to the two-phase nature of the flow problem and the importance of mixing the two phases in each other, the Eulerian-Eulerian approach is used to model continuous and dispersed phases. In this study, multiPhaseEulerFoam solver has been used.

D. Geometry, Meshing and Boundary Condition and Grid Independence

In the present study, various configurations are introduced, including the angles and heights of the ramps. To construct the geometry of the numerical model, a code is written in the FORTRAN code. An input data is formed where it includes the ramp angle and height, the distance to the ramp entrance and the distance from the end of the ramp. The information needed for constructing the geometry of flow field in the OpenFOAM software is extracted from the FORTRAN code and transferred to BlockMesh environment inside the OpenFOAM software. Owing to the dimensions of the study zone and air bubble diameter, the geometry of the numerical model is prepared using a grid size of 5 mm. The meshing of the flow field considered in the numerical model is demonstrated in Figure 5. The sensitivity analysis of the numerical model is conducted concerning the mesh size. It is found that using a cell size smaller than 5 mm will not promote the accuracy of the calculations. In experiments, the air is injected with different discharge rates through the air duct into the air cavity formed immediately downstream of the ramp. We have considered a similar procedure in the numerical model to verify the numerical model. Simulations of this study are prepared in a two-dimensional time-dependent domain using the OpenFOAM software.

Zhang compared the performance of the Standard K-ε and RNG K-ε models applied for aerated flow. The results indicated that there are slight differences between the two models. Nevertheless, the RNG k-ε turbulent model is more appropriate to represent characteristics of turbulent free-surface flow [16]. Therefore, the two-equation turbulence RNG k-ε model is applied to a numerical model in this research.
In this research, the following boundary conditions are used for the numerical model: 1- water inlet: mass flow rate 2- air inlet: mass flow rate 3- outlet: pressure outlet 4- channel body such as the floor, ramp and side walls: wall, 5- channel roof: atmosphere.

III. RESULT AND DISCUSSION

The numerical model’s performance is verified using the experimental results of cavity length formed immediately downstream of the ramp, and the mean flow velocity. In the next step, the flow characteristics along the chutes such as velocity, air concentration, turbulent intensity and water surface profile for some runs are numerically extracted and discussed where a broad understanding of flow behaviour around the aerator ramp has been achieved.

A. Verification of Numerical Model

1) Cavity Length \( L_c \): The deflector separates water flow passing over the aerator ramp from the chute bottom, and a cavity is formed immediately downstream of the ramp. As a result, airflow is forcefully drawn into the cavity. The cavity length, marked as \( L_c \) (m), is the distance from the aerator ramp to the point where the jet hits the floor of the chute (reattachment point \( R \)) (Figure 6). To delineation the air-entrainment capacity of an aerator, coefficient \( \beta \) is defined as the ratio of airflow discharge \( Q_a, \text{m}^3/\text{s} \) to water flow discharge \( Q_w, \text{m}^3/\text{s} \).

Figure 7 demonstrates the comparison of the cavity lengths \( L_c \) gained from the numerical and laboratory models for different runs. For all runs, the cavity lengths are estimated by the two-dimensional numerical model more than the laboratory model. The difference percentages calculated for the two models’ cavity lengths are less than 8%, confirming that the simulation carried out by the numerical model is satisfactory.

2) Mean velocity: The mean velocity distribution profiles obtained from the numerical and laboratory models in the flow direction is displayed in Figure 8 on the ramp tip (Section \( S_1 \) seen in Figure 3) for runs 3 and 5 \((L=4\text{cm}, \theta=10^\circ)\). The conformity of numerical and experimental results implies that the numerical model appropriately simulates the aerator ramp’s flow velocity field. As shown in Fig. 9, the flow velocity gradually increased from the bed bottom to reaching a threshold value.

B. Variation of Flow Characteristics around the Aerator Ramp

In order to better understanding the flow field in the vicinity of the aerator ramp, variations of flow characteristics such as velocity, air concentration, turbulent intensity, and water surface are investigated. Profiles of these characteristics obtained from the numerical model are presented in various sections along the chute. Coordinate of these 12 sections is displayed in Table II as well as Figure 10. Furthermore, the effect of the Froude number of approaching flow is studied on the turbulence intensity and aeration coefficient of flow passing the aerator ramp.

| TABLE II | LOCATION OF THE SECTIONS ALONG THE CHUTE |
|----------|------------------------------------------|
| Section  | \( S_1 \) | \( S_2 \) | \( S_3 \) | \( S_4 \) | \( S_4C \) | \( S_5 \) | \( S_6 \) |
| \( X(\text{m}) \) | 1.54 | 1.84 | 2.04 | 2.20 | 2.23 | 2.32 | 2.44 |

| Section  | \( S_7 \) | \( S_8 \) | \( S_9 \) | \( S_{10} \) | \( S_{11} \) | \( S_{12} \) |
|----------|--------|--------|--------|-----------|-----------|--------|
| \( X(\text{m}) \) | 2.74   | 3.04   | 3.5    | 4.0       | 4.5       | 4.8    |
1) **Velocity Profiles**: Velocity Profiles are presented in Figure 10 at 12 sections along the chute for Run8 (t=4cm, θ=10°, φ=14.5°) where θ and φ are angles of the ramp and chute bed, respectively. As shown in Figure 11, in all profiles (before the ramp, over and after the ramp), the flow velocity gradually increased from the minimum value on the floor to a threshold value. Velocity profiles at upstream of the ramp (S₁, S₂), on the tip of the ramp (S₃), and shortly thereafter (S₄, S₄C), on cavity region (S₅, S₆, S₇), near reattachment point (S₈), and downstream region of reattachment point (S₉, S₁₀, S₁₁, S₁₂) have a variety of forms depending on the hydraulic and geometrical conditions.

As seen (Figure 11) from profile S₈ near the jet collision region, the flow velocity near the floor is significantly reduced compared to the other profiles due to the jet collision with the floor.

2) **Air Concentration Profiles**: In this simulation, the flow field is aerated by two mechanisms: 1) natural entrainment from upper nappe of jet; 2) from lower nappe of the jet in which aerated top boundary of the cavity where a strong turbulent happens. The percentage of aeration from the lower surface of the jet at the reattachment point is considerable due to the higher level of turbulence. Figure 12 shows the C distribution as a function of the relative depth of flow (Y/Y₉₀), in which Y is water depth, and Y₉₀ is water depth where C=90% at different sections.

As seen from Figure 12, at section S₁ located upstream of the ramp aerator, there is no air in the lower layer of water flow close to the chute bed. Air is gradually entrained into the flow-through free surface. S₁ to S₇ placed in the cavity zone downstream of the ramp, C, which is maximum inside the cavity, is gradually reduced by moving upward in the water body until half of the water depth where air concentration is reached to a minimum value. After that C is progressively increased to a maximum amount on the free surface of the water. Downstream of the cavity zone, the de-aeration process in flow direction causes air bubbles to escape through the free surface; thus, uniform flow is formed with a steady distribution of C, as seen in section S₁₁ of Figure 12. The variation pattern of C obtained here in the vicinity of the aerator ramp adapts similar research results found in the literature [13]–[15].

---

**Fig. 10** Location of the sections along the chute

**Fig. 11** Velocity distribution profiles at different sections around the chute aerator
3) **Water Surface Profile:** The water surface profile along the chute for Run 8 is presented in Figure 13. It is seen that water depth which is 9cm at the chute inlet, decreases slowly till the position of the ramp. It increases slightly over the ramp and fluctuates at the reattachment zone. Thereafter it reduces progressively till the end of the chute. On the contrary, the flow velocity increases slowly along the chute.
4) Turbulent Intensity Profiles: Figure 14 shows the distribution of Turbulent Intensity ($Tu = u'/U$) at various vertical sections around the aerator ramp. Where $u'$ and $U$ are velocity fluctuation and mean velocity, respectively. Also, $Z$ is the height from the bottom of the flow body, and $h$ is the water depth. As seen from Figure 14, the turbulent intensity along the aerator ramp is increased from small amounts at the beginning of the ramp (Section S2) to maximum values at the end of the ramp (Section S3); therefore ramp acts as a turbulent generator. After the ramp, the turbulent intensity gradually decreased along the chute.

5) Flow Turbulent Intensity versus Froude Number: To investigate the variation of the turbulent intensity parameter of flow against Froude number, simulations are carried out for flows of 5 different Froude numbers ranging from 5.5 to 8.5. The profiles of the turbulent intensity distribution are demonstrated at section $S_3$ positioned on the aerator tip for various Froude numbers (Figure 15).

The results indicate that the rate of turbulent intensity is relatively increased when the Froude number grows. For a constant Froude number, the turbulent intensity gradually increases from the ramp tip up to a maximum value at the level of 6-8% of flow depth and thereafter, it reduces to a minimum amount close to the water-free surface, as seen from Figure (15a, b).

6) Aeration Coefficient ($\beta$) versus Froude Number: the air-entrainment capacity of an aerator that is expressed as an aeration coefficient ($\beta$) is defined as the ratio of air discharge ($Q_{air}$, m$^3$/s) to water flow discharge ($Q_w$, m$^3$/s). In order to calculate accurately the amount of air entering the flow through the aerator system mounted on the chute bed, the airflow is measured at the sections before ramp ($S_2$) and after the reattachment point ($S_9$). The differences between these sections’ obtained values indicate the exact amount of air entering by the aerator into the flow. Figure 16 shows how the air discharge ($Q_{air}$, m$^3$/s) and aeration coefficient ($\beta$) change by increasing the Froude number from 5.5 to 8.5. These Froude numbers correspond to 30, 50, 70, 90, 110 and 130 lit/s water discharge ($Q_w$). The results show that the air-entrainment coefficient ($\beta$) increases linearly with the increase of the Froude number.

IV. CONCLUSION

The results of this numerical investigation on the flow field around the chute aerator using OpenFOAM open-source software can be summarized as follows: Verification of results indicates a proper agreement between numerical and experimental results so that the difference between the numerical and laboratory results for the cavity length is, on average, 5% for all performances. R-squared results are 0.958 and 0.93 for the velocity result of RUN 3 and 5, respectively. Also, the modeling capability of the multiPhaseEulerFoam solver is justified by simulating the air-water flow around the chute aerator. Velocity increased gradually from the chute bed and reached maximum value on the near water surface. As the Froude number increases, the turbulence intensity profiles at the tip of the ramp show an increasing trend; thus, the ramp acts as a turbulent generator. The maximum value of turbulence intensity occurs at 6-8% of flow depth ($h$) on the tip of the ramp. As
the Froude number increases, both air flow discharge \((Q_a)\) and aeration coefficient \((\beta)\) increase linearly.

| NOMENCLATURE |
|-----------------|
| \(h\) | water depth \(m\) |
| \(Q_{air}\) | air discharge \(m^3s^{-1}\) |
| \(Q_w\) | water discharge \(m^3s^{-1}\) |
| \(Y_{90}\) | water depth where C=90% \(m\) |
| \(T_o\) | turbulent intensity |
| \(L_c\) | cavity length \(m\) |
| \(u'\) | velocity fluctuation \(ms^{-2}\) |
| \(U\) | mean velocity \(ms^{-2}\) |
| \(Z\) | height from the bottom of water body \(m\) |

Greek letters

| \(\theta\) | aerator ramp angle \(deg\) |
| \(\beta\) | aeration coefficient |
| \(\rho\) | water density \(kg.m^{-3}\) |
| \(\varphi\) | chute angle \(deg\) |

REFERENCES

[1] H. Falvey, “Cavitation in Chutes and Spillways” Engineering Monograph 42, United States Department of the Interior, Bureau of Reclamation, Denver, Colo, USA. 1990.

[2] J. A. Kells, and C. D. Smith, “Reduction of cavitation on spillways by induced air entrainment” Canadian Journal of Civil Engineering, Vol 18, pp. 358–377, 1991.

[3] N. L. Pinto, “Prototype aerator measurements” IAHR Hydraulic Structures Design Manual No. 4, Ed. I.R, 1991.

[4] W. Wei, J, Deng, and Z. Faxing “Development of self-aeration process for supercritical chute flows” International Journal of Multiphase Flow, Vol 79, pp. 172–180, 2016.

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

[6] J. M. Zhang, J. G. Chen, and W. L. Xu, “Three-dimensional numerical simulation of aerated flows downstream sudden fall aerator expansion in a tunnel” Journal of Hydrodynamics, Vol 23(1), pp. 71–80, 2011.

[7] R.W.P. May, P. M. Brown, and I. R. Willoughby, “Physical and numerical modeling of aerators for dam spillways” Hydraulics Research Report, No. SR 278, Wallingford, U.K. 1991.

[8] D. Vischer, P. Volkart, and A. Sigenthaler, “Hydraulic modelling of air slots in open chute spillways” International Conference on the Hydraulic Modelling of Civil Engineering Structures, 1982, Coventry, U.K., pp. 22–24.

[9] J. Yang, P. Teng, and H. Zhang, “Experiments and CFD modeling of high-velocity two-phase flows in a large chute aerator facility” Journal of Engineering Applications of Computational Fluid Mechanics, Vol 13(1), pp. 48-66, 2018.

[10] H. Chanson, “Study of Air Entrainment and Aeration Devices” Journal of Hydro Research, IAHR, Vol 27 (3), pp. 301-319, 1989a.

[11] H. Chanson, “Aeration of a free jet above a spillway” Journal of Hyd. Res., IAHR, Vol 29(6), p. 864, 1991.

[12] I. Schwartz, and L. P. Nutt, “Projected nappes subjected to transverse pressure” Journal of the Hydraulics Division, Vol 89(7), pp. 97–104,1963.