Proper Numerical simulation of water surface profile over stepped spillway Using Different Solvers

F. Hatam[1], M.M. Alishahi[2], A.R. Keshavarzi[3]

1- Graduated student of Water Engineering Department, Shiraz University
2- Professor of School of Mechanical Engineering, Shiraz University
3- Professor of Water Engineering Department, Shiraz University

fateme.hatam@gmail.com

Abstract

Enhancing the energy dissipation along a spillway is of great importance for more efficient design of the downstream structures. In this regard, a number of steps can be added to the spillway in order to increase the flow turbulence. By this action the dimensions of stilling basin and consequently the cost is significantly reduced. In this research the flow over stepped spillway and the downstream of hydraulic jump is modeled using FLUENT and FLOW-3D softwares. The water surface profile over the spillway and the hydraulic jump is then compared with that of the experimental model and good agreement is observed. The capability of FLOW-3D in allowing the air to escape through free surface is investigated by simulating a dam break problem and comparing it with the existing experimental result. The results show that this software allows the air escape through a free surface if the Drift-flux model and gas escape model is invoked.

Key words: Spillway, Hydraulic jump, water surface profile, FLUENT, FLOW-3D

1. Introduction

Spillways are a kind of hydraulic structures that are used to convey the water when the reservoir capacity is less than the difference between the volumes of inflow and outflow. Spillways can be classified into different groups such as Ogee spillway, Chute spillway, Side channel spillway, Shaft spillway, Siphon spillway and stepped spillway.

Stepped spillway has been used over 3000 years ago. In recent years high attention is given to this kind of spillway. In 19 and 20 centuries many of the spillways were constructed as stepped spillway, however before this stage, because of development in designing and constructing of stilling basin, using stepped spillway had been outdated for a period of time. But recent progress in new technology such as roller compacted concrete (RCC) caused a new attention toward stepped spillway.

The flow over a stepped spillway could occur in three different types: nape, transition and skimming flow. Therefore, the investigation of hydraulic behavior of the flow over this kind of spillway becomes complicated. On the other hand, because of the air entrainment, turbulence and high interaction between air and water, accurate flow understanding and visualization is required for safe and reliable design of stepped spillway..

The spillway should pass the designed flow without causing any damage to the structure and the surrounding environment. Water energy should be properly dissipated to avoid dangerous damage at the toe and downstream of the spillway. One choice is to establish a stilling basin at the downstream of the spillway. There are different types of energy dissipaters that some of them are as follows; stilling basin type I, type II, type III, type IV, Solid Bucket and Slotted Bucket. The type of stilling basin is chosen according to the hydraulic characteristics of flow as well as economical and surrounding conditions. Another method is to dissipate the energy along the spillway by adding appropriate type and number of steps on the spillway to intensify turbulence of the flow and consequently increasing the energy dissipation over the spillway. In this condition, the dimension and the cost of downstream dissipater structure will be reduced.

In 2002 Chen et. al claimed that they had numerically simulated *turbulent* flow over stepped spillway for the first time. They applied k-ε turbulent model in their simulation [1]. In 2005 Tabbara et. al simulated the flow over stepped spillway without considering the air entrainment phenomenon, a finite element based software (ADINA) was used in their research [2]. Cheng et. al tried to model the air water flow in 2006. They applied finite volume approach of a mixture multiphase model to simulate the flow over stepped spillway [3]. In another research in 2006, the flow was modeled using VOF[1] scheme over stepped channel by Lee and Dong [4]. Numerical simulation of flow over the stepped spillway was conducted using mixture multiphase model by ZhongDong et. al in 2009. In this research four different turbulence models were chosen to solve the related equations. They found that realizable k-ε model is the most efficient turbulent model in flow simulation over stepped spillways [5]. In 2009, Carvalho and Martins investigated the flow over the steps of a stepped spillway using analytical, physical and numerical methods. Experimental modeling was studied to minimize the hydraulic jump length, maximizing discharge per unit width and examine resemblance of hydraulic jumps on each step. The numerical method using VOF scheme and RNG k- ε turbulent model was applied to evaluate the velocity, pressure and hydrodynamic forces on the sill [6]. In 2010, a dam break analysis using MPS[2] method and an experimental model was done by Hu and Sueyoshi [7].

In the present research, in order to compare the capabilities of different solvers in simulating the flow over stepped spillway and downstream hydraulic jump, FLUENT and FLOW-3D commercial softwares were used. Water surface profile was then compared with the experimental model which was built in the laboratory. In the next part, the capability of FLOW-3D, in allowing the air to escape through the free surface was investigated. This investigation is carried out because of the importance of this fact in a stepped spillway study. This effect is more dominate at the end of the spillway, where no more steps exist to produce turbulence and air entrainment into the flow. In the experimental conditions, because of the restriction in discharge and the dimensions of the model in the laboratory, air entrainment does not exist in flow over the spillway. Accordingly, for testing the mentioned phenomenon, a dam break problem was chosen and the results are compared with existing experimental data (Sueyosho and Hu, 2010).

2. Numerical simulation

In this paper FLOW-3D and FLUENT are employed to simulate the flow over the spillway and downstream hydraulic jump. VOF and TruVOF methods are used in FLUENT and FLOW-3D respectively to compute the hydraulic jump and the water surface profile along the spillway and the results are compared with the experimental data. The spillway located in a flume in the laboratory located in the Agricultural School of Shiraz University. The height, width and length of the flume are 60 cm, 70 cm and 15 m respectively. The stepped spillway has 6 steps and the height of the first five steps is 6.5 cm and the last step height is 7 cm. the first step length is 19 cm and others are 16 cm length (Fig.1). The geometry creation and mesh generation is done using GAMBIT and generated data imported to FLUENT. However in FLOW-3D simulation, the geometry is produced by AutoCAD and mesh generation is done by FLOW-3D.

To control the location of the hydraulic jump at the downstream of the spillway, a slide gate is used at the end of the channel to adjust the tail water depth. In this experiment the tail water is adjusted to 25 cm, and the discharge is 107 L/s. The main goal of this experiment is just to compare the water surface profile and downstream hydraulic jump with numerical simulations. Therefore, the flow depth at the upstream and downstream of hydraulic jump, the location of hydraulic jump and the water surface profile are measured in this experiment.

2.1. Governing Equations and Numerical Methods

The governing equations include continuity and momentum equations for the incompressible multiphase fluids. The continuity equation is as follows (streeter 1961);

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho\vec{v}) = 0 \qquad (1)$$

In which v is the mixture velocity, v is defined as: $v = \alpha_w v_w + (1-\alpha_w)v_a$ where v_w and v_a are the velocity of water and air, respectively. ρ is the mixture density, ρ is defined as: $\rho = \alpha_w\rho_w + (1-\alpha_w)\rho_a$ where ρ_w and ρ_a are the density and α_w and α_a are the volume fraction of water and air, respectively.

The momentum equation in this model is a single equation for the whole domain of air and water represented as follows:

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla.(\rho\vec{v}\vec{v}) = \nabla p + \nabla.[\mu(\nabla\vec{v} + \nabla\vec{v}^{T})] + \rho\vec{g} + \vec{F} \qquad (2)$$

---

[1] Volume Of Fluid

[2] Moving Particle Semi-implicit

Where p is pressure, μ=μ0+μt and μ0 are air-water mixed viscosity. For all simulations in this paper, RNG k-ε turbulent model is used to solve the turbulent flow over spillway.
FLUENT is developed based on finite volume numerical discretization technique. One of the available multiphase approaches in FLUENT is VOF model. Some applications of VOF scheme mentioned by the software developers are as follows: free surface flows, the motion of large bubbles, dam break modeling and finding any liquid-gas interface [9]. The VOF scheme is a surface tracking technique which is applied to a fixed Eulerian grid. This scheme can model two or more immiscible fluids by solving one set of momentum equations and tracking the volume fraction of each fluid within any computational cell in the whole domain.
If volume fraction of q_{th} fluid in the cell is named as α_q then three states exist as follow:
α_q=0, the cell is empty of the q_{th} fluid,
α_q=1, the cell is full with the q_{th} fluid,
$0< \alpha_q <1$, the cell is the interface between q_{th} fluid and one or more fluids.
Tracking the interface between the phases is done by solving continuity equation for the volume fraction of one or more the phases. For phase q_{th} this equation is as follows:

$$\frac{1}{\rho_q}\left[\frac{\partial}{\partial t}(\alpha_q\alpha_q)+\nabla.\left(\alpha_q\rho_q\vec{v}_q\right)=S_{\alpha_q}+\sum_{\rho=1}^{n}(\dot{m}_{\rho q}-\dot{m}_{q\rho})\right] \tag{3}$$

In which $\dot{m}_{qp}$ and $\dot{m}_{pq}$ are the mass transfer from phase q to p and p to q, respectively. By default the right term of the equation (source term) is equal to zero but this can be changed for each phase. The volume fraction equation is not solved for the primary phase. This volume fraction is computed using the following equation:

$$\sum_{q=1}^{n}\alpha_q=1 \tag{4}$$

The equations in FLOW-3D software are similarly based on the momentum and continuity equations. When simulating to determine the interface between different components, VOF scheme is also applied by the software.

2.2. Boundary conditions

The boundary conditions that are used in FLUENT are illustrated in Figure 1.

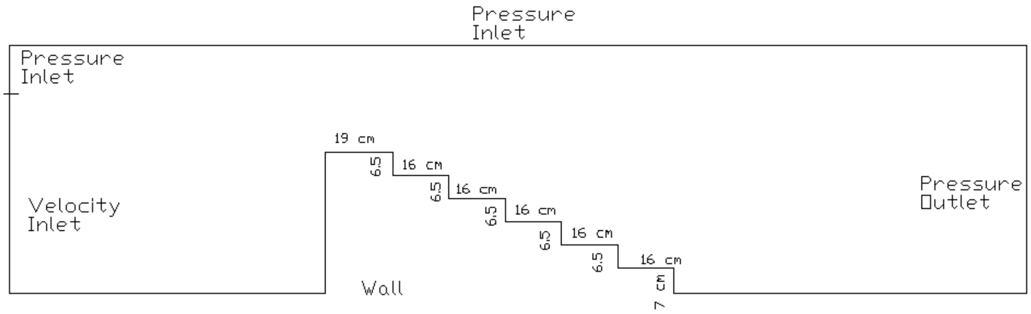


Figure 1. Boundary conditions which are used in FLUENT.

The boundary conditions that are used in FLOW-3D are observed in Figure 2. where P means pressure boundary condition. In this model the hydrostatic pressure is defined according to the tail water depth. Q is an inlet discharge boundary condition, W is a wall boundary condition and S is a symmetric boundary condition (a boundary with zero flow passing through it and no stress).

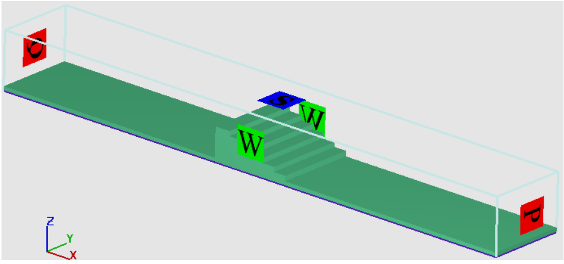
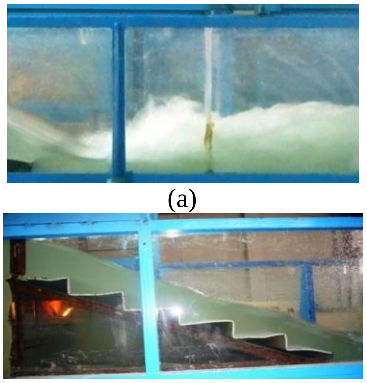


Figure 2. Boundary conditions which are used in FLOW-3D.

3. Results

3.1. Example 1

The flow over the spillway and the downstream hydraulic jump of the experimental model is shown in Figure 3. The spillway geometry and the mesh generation is created using the aforementioned software. Grid independent solution is approximately achieved with a 600 by 90 grid in 2D case. It must be mentioned that the discretized domain before and after the spillway is 3 m. In Figs. 4 and 6 the water surface profiles that are resulted from the numerical simulations, are compared with that of experimental model and as it is observed, good agreements are achieved. As it is shown in Figure 5., FLOW-3D was also able to model the water surface oscillations at downstream of the hydraulic jump as observed in experimental model .


(a)

(b)

Figure 3. Flow over the spillway and the downstream hydraulic jump of experimental model.

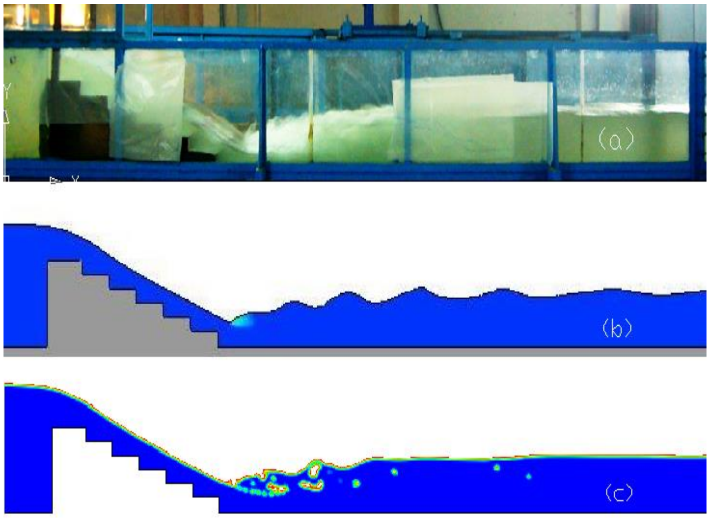


Figure 4. Water surface profile (a) experimental model (b) FLOW-3D (c) FLUENT.

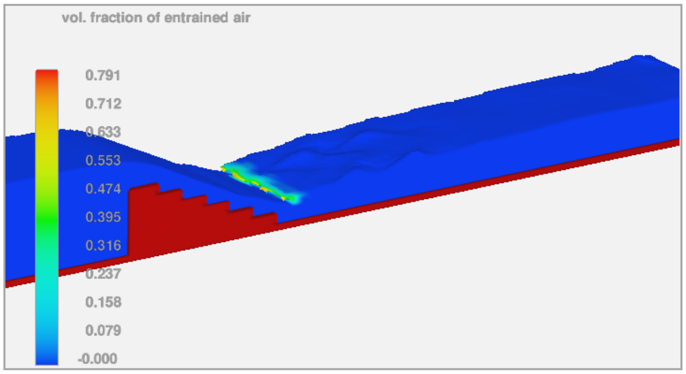


Figure 5. Isometric view of three dimensional flow simulation over spillway (FLOW-3D).

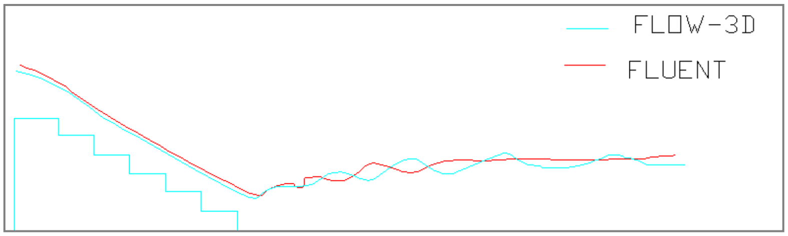


Figure 6. Comparison of water surface profiles by different softwares.

Figure 7 shows the velocity magnitude contours derived by numerical simulations. The values demonstrate that 68 percent of the energy is dissipated along the spillway and the rest is dissipated by the hydraulic jump and along the channel. This value is computed from $E = \frac{V^2}{2g} + z$ in which V is the depth average velocity, z is the elevation and E is the flow energy in each section.

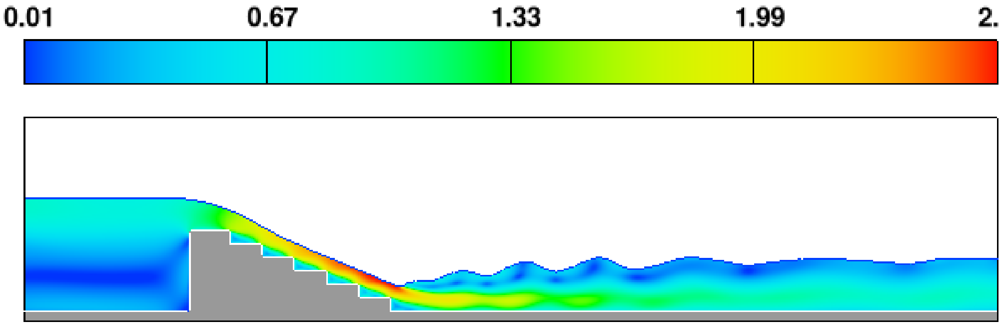


Figure 7. Velocity magnitude resulted from numerical simulation (using Flow-3D).

3.2. Example 2: Dam Break

It is noted that any composition of the mixed flow may be composed of different components such as fluid-solid and fluid-bubbles (like the flow on stepped spillway or/and hydraulic jump) in which any component with different density can have different velocities. Density variation cause non uniform body force and this fact cause different velocities. In most cases the differences in velocity is really significant such as movement of gravel in water; however there are also some cases such as moving silt in water that the relative phase velocities are very small. By using the concept of slip velocity the model allows the various phases to move at different velocities with respect to each other. This subject becomes important when the particle size of the secondary phase is as large as its weight or lift force to be significant or/and the volume fraction of the secondary phase is comparable with main flow. In these conditions some of the forces acting on the particle such as gravity or lift and the interaction between the particles become important. Therefore the secondary phase will move with a different velocity from the main flow.

The relative velocity between the suspended phase and continues phase can be computed from the following equations:

$$u_r = \left(\frac{V_P}{K_P}\right)\frac{f(\rho_2-\rho_1)}{\bar{\rho}}\nabla P \qquad (5)$$

In this equation it is assumed that the suspended phase is composed of particles with the same size and n is the number of particles which are located in a unit volume. Volume of particle (V_p) is $(1-f)/n$, k_p is the drag coefficient and the volume weighted average density is represented as follow, [10].

$$\bar{\rho} = f\rho_1 + (1-f)\rho_2$$

Here a dam break example is solved to investigate the effect of Drift-flux model to examine the capability of FLOW-3D in allowing the air to escape through a free surface. It is a special case of using the Drift-flux model. To validate the result of numerical simulation the experimental data of Hu and Sueyoshi (2010) is used. The water tank dimensions are illustrated in Figure 8.

Figure 9 (a) shows the condition that the drift-flux model is not applied and consequently the air has not been allowed to escape through the free surface during the simulation. In this case, although the air enters into the flow by sudden and rapid motion of water, but it is not able to escape through the free surface when the flow become nearly steady while it should physically exit from the flow as it is observed in physical model. At the end of simulation the flow depth is determined with the sum of the volume of the water and the entered air during the water motion. This phenomenon certainly causes error in the simulation result. However by applying the Drift-flux model and also allowing the air to escape through the free surface, it is observed that the air escapes through the free surface correctly, when the flow is becoming steady during the simulation. This concept can be seen in Fig.9 column (c).

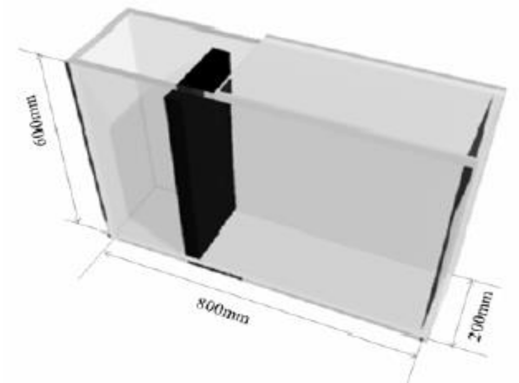


Figure 8. Water tank dimensions Hu and Sueyoshi (2010).

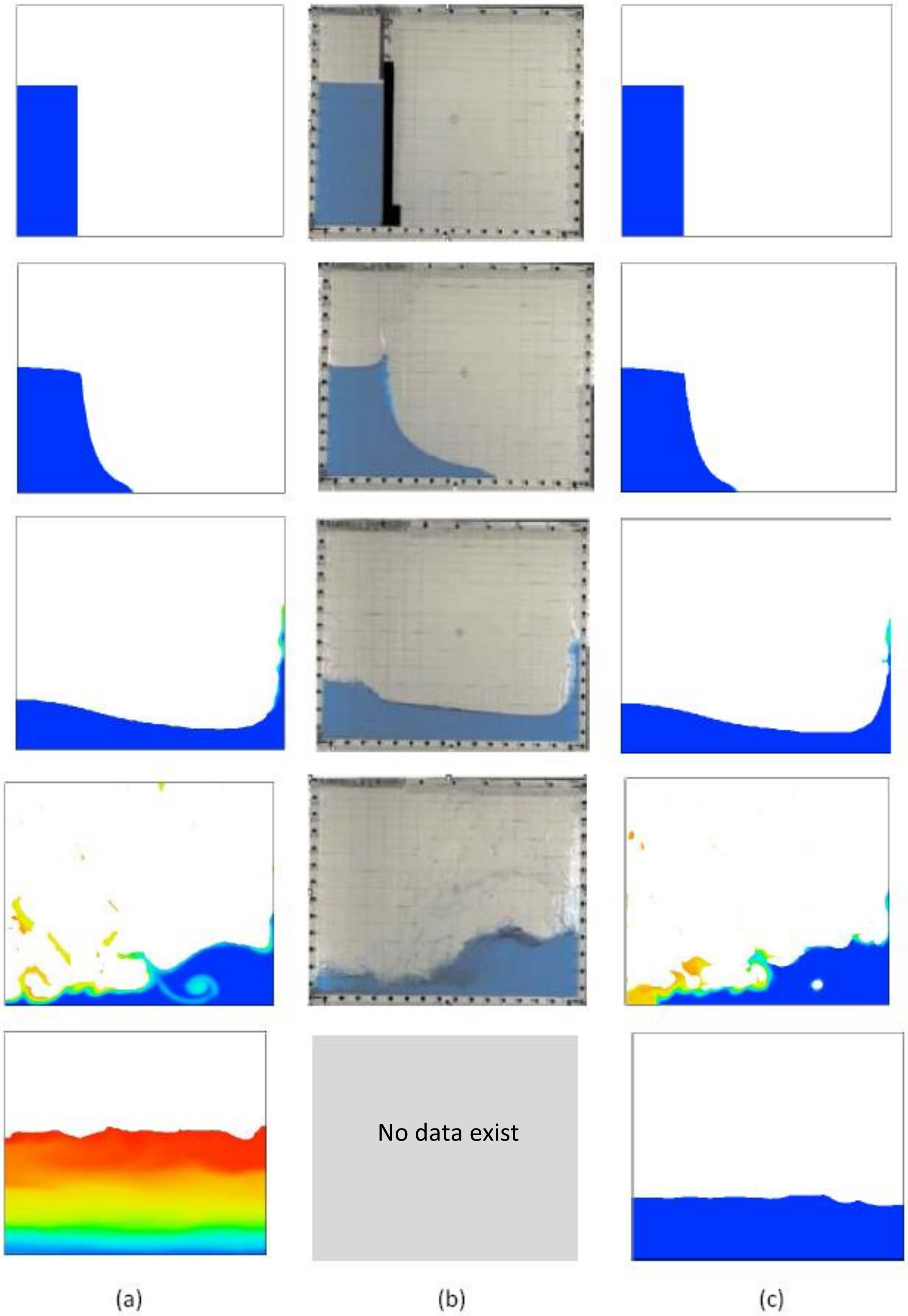


Figure 9. Dam break simulation (a) without applying Drift-flux model (b) experimental results (Hu and Sueyoshi, 2010) (c) applying Drift-flux model and allowing the air to escape through the free surface.

In order to valuate Fluent capability to simulate air escape from the water one may have to use appropriate multiphase approach including suitable particle size and slip velocity parameters. Based on the author's experience, Mixture and Eulerian models are potentially capable to do this job in FLUENT. This evaluation needs to be investigated in further research.

4. Conclusion

In this research the flow over stepped spillway and downstream hydraulic jump is simulated using FLUENT and FLOW-3D. Water surface profile is then compared with the experimental model built in the laboratory. Good agreement is achieved between numerical results and experimental model. As it is seen in three dimensional simulation, FLOW-3D is also able to simulate the water surface oscillations at the downstream of hydraulic jump as observed in experimental model too.

The capability of FLOW-3D in allowing the air to escape through the free surface was also investigated. This effect is the governing factor at the end of the spillway, where there is no steps to produce turbulence or air entrainment into the flow. In this study, because of some restrictions in discharge and dimensions of the laboratory model, air entrainment does not exist in flow over the spillway. Therefore to test the mentioned phenomenon, a dam break analysis is carried out and the results are compared with existing experimental data (Sueyosho and Hu, 2010). It is observed that the numerical model developed in FLOW-3D can allow the gas to escape from the free surface and therefore preserve the correct volume of the fluid at final stage.

5. References

[1] Chen, Q., Dai, G., Liu, H. (2002). “Volume of Fluid Model for Turbulence Numerical Simulation of Stepped Spillway Overflow.” J. Hydraul. Eng., 128(7), pp. 683-688.

[2] Tabbara, M., Chatila, J., Awwad, R. (2005). “Computational simulation of flow over stepped spillways.” Comput. Struct., 83, pp. 2215–2224

[3] Cheng, X., Chen, Y., Luo, L. (2006). “Numerical simulation of air-water two-phase flow over stepped spillway.” Sci. China Ser. E: Technol. Sci., 49(6), pp. 674-684.

[4] Dong Z.Y., Lee J. H.W. (2006). “Numerical simulation of skimming flow over mild stepped channel.” J. Hydrodyn. Ser. B., 18(3), pp. 367-371.

[5] ZhongDong, Q., XiaoQing, H., WenXin, H., António, A. (2009). “Numerical simulation and analysis of water flow over stepped spillways.” Sci. China Ser. E: Technol. Sci., 52(7), pp. 1958-1965

[6] Carvalho, R. F., Martins, R. (2009). “Stepped Spillway with Hydraulic Jumps: Application of a Numerical Model to a Scale Model of a Conceptual Prototype.” technical note, J. Hydraul. Eng., 135(7), pp. 615-619.

[7] Hu, C., Sueyoshi, M. (2010). “Numerical Simulation and Experiment on Dam Break Problem.” J. Marine. Sci. Appl., 9, pp. 109-114.

[8] Streeter, V.L., 1961. Handbook of Fluid Dynamics. McGraw-Hill Inc. U.S. Army Corps of Engineers, 1964. Air-demand Regulated Outlet Works. Hydraulic Design Criteria, Chart 050 -1.

[9] FLUENT Inc., (2006). FLUENT v. 6.3 user’s guide, Lebanon, NH.

[10] Flow Science, Incorporated, (2008), “FLOW-3D Users Manual Version 9.3”, Santa Fe, New Mexico.