INTERNATIONAL JOURNAL OF
ENERGY AND ENVIRONMENT

Volume 6, Issue 6, 2015 pp.597-606

Journal homepage: www.IJEE.IEEFoundation.org

Stepped spillway optimization through numerical and physical modeling

Hamed Sarkardeh[1], Morteza Marosi[2], Raza Roshan[2]

[1] Department of Engineering, Hakim Sabzevari University, Sabzevar, Iran.
[2] Hydraulic Structures Division, Water Research Institute (WRI), Hakymia, Tehran, Iran.

Abstract
The spillway is among the most important structures of a dam. It is importance for the spillway to be designed properly and passes flood flow safely with more energy dissipation. The zone which ogee spillway crest and stepped chute profile are joined with each other is important in design view. In the present study, a physical model as well as a numerical model was employed on a case study of stepped spillway to modify the transitional zone and improve flow pattern over the spillway. Many alternatives were examined and optimized. Finally, the performance of the selected alternative was checked for different flow conditions, air entrainment and energy dissipation. To simulate the turbulence phenomenon, RNG model and for free surface VOF model was selected in the numerical model. Results of the numerical and physical models were compared and good agreement concluded in flow conditions and energy dissipation.

Keywords: Stepped spillway; Physical and numerical simulation; Flow pattern; Air entrainment; Energy dissipation.

1. Introduction
Stepped spillways construct over chute with sloping floors and can be used to convey floods at high dams. They are found to be effective for dissipating energy of excess flood released from dams passed over the steps. Many studies have shown that favorable design of stepped spillways can decrease the size of the stilling basin significantly and thus saving on construction costs [1, 2]. Stepped spillways have gained much interest in recent decades because of their compatibility with Roller Compacted Concrete (RCC) dams, hence having a steep slope. Once a stepped spillway is located on the body of a RCC dam, it has additional advantages in construction and economic. Depending on the flow discharge for certain stepped spillway geometry, the flow over the steps could be divided into three distinct flow regimes: nappe, transition and skimming flow by increasing flow discharges. Improving in design parameters of a stepped spillway could perform by both numerical and experimental simulations. Both numerical and experimental modeling are important and seems be necessary to verify each other in design of infrastructures as well as dams.
Numerical modeling techniques have been developed in a wide range of engineering application in the recent years and were used widely to simulate flow conditions over spillways. Song and Zhou used Large Eddy Simulation (LES) in combination with an explicit finite volume scheme to determine the flow over an ogee overflow spillway [3]. They compared time averaged results of the numerical and physical

models. The findings of this study show that both numerical and physical models are in good agreement. Savage and Johnson computed discharge and crest pressures over an uncontrolled USACE and USBR standard ogee crested spillway using Flow 3D Software [4]. Results of their numerical study were compared with experimental and USACE & USBR data. It was found that the computed discharges by Flow 3D Software were placed between the experimental study and USACE & USBR data. Ho et al. made a comparison on crest pressures and discharges over a standard ogee spillway by 2D and 3D simulations in Flow 3D Software and USACE data and also empirical discharge equations [5]. Gessler documented how Flow 3D Software was used to model discharge over an overflow spillway with newly computed probable maximum flood levels [6]. Fabian et al. presented the results of a comprehensive study on stepped spillways numerically and experimentally [7]. Some studies also were focused on air entrainment in stepped spillways [8, 9]. In this paper modification of a stepped spillway are presented by both physical and numerical models. Chen et al. [10] used the $k-\in$ turbulence model to simulate the complex turbulence overflow. Their first five steps were varied while the sizes of the rest were 0.06m high and 0.045m long. The study indicated that the turbulent numerical simulation is an efficient and useful method for the complex stepped spillway overflow.

Usual method for joining two different parts (crest and chute) of a stepped spillway is attachment them directly. This method may cause jumping the flow in low discharge and therefore in some cases, it is necessary to change geometry of first steps to improve flow condition. Moreover, flow pattern and energy dissipation over stepped spillways were investigated to be a guideline in new spillway design.

2. Physical model

In the present research, the physical model of Zhaveh spillway was used. The Zhaveh Dam is located in Kordestan Province in west of Iran with height of 85m. Zhaveh Dam is equipped with a 55m wide stepped spillway for releasing design flood of about 1000m^3/s. A schematic view of dam body, stepped spillway and reservoir are shown in Figure 1.

By considering all scale effects in physical modeling, the 1:25 scale was selected and physical model was constructed. In this model general slope of spillway was 1.2V:1H and all steps have 1.2m height and 1.0m length. Sharp crested rectangular spillway was installed downstream of the model and were used for discharge measurement. Also, a limnimeter with 0.1mm accurate was used to measure elevation of water surface. Pressure measurements were carried out in different points along the spillway and steps. Velocities also were measured by means of a propeller along the spillway. To see the flow pattern and regime, all side walls and bed of stepped spillway were made from Perspex (Figure 2).

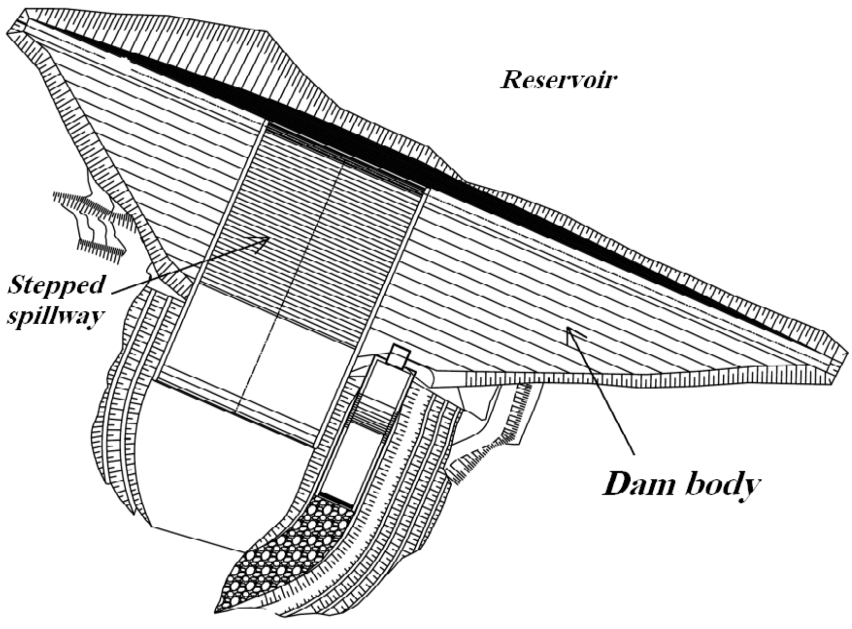


Figure 1. Schematic view of Zhaveh Dam

Figure 2. Constructed physical model

3. Numerical model

Flow 3D Software which is one of the powerful numerical modeling software capable of solving a wide range of fluid flow problems, was used to simulate flow over the stepped spillway. The Volume Of Fluid (VOF), Fractional Area/Volume Obstacle Representation (FAVOR) and the renormalization group (RNG) were implemented to simulate the fluid surface, obstacles and turbulence. The continuity and momentum equations for flow and transport equation for VOF are outlined in (1) and (2) and (3):

$$\frac{V_f}{\rho}\frac{\partial\rho}{\partial t}+\frac{1}{\rho}\nabla.\left(\rho\vec{u}A_f\right)=-\frac{\partial V_f}{\partial t} \quad (1)$$

$$\frac{\partial\vec{u}}{\partial t}+\frac{1}{V_f}\left(\vec{u}A_f.\nabla\vec{u}\right)=-\frac{1}{\rho}\left[\nabla P+\nabla.\left(\tau A_f\right)\right]+\vec{G} \quad (2)$$

$$\frac{\partial F}{\partial t}+\frac{1}{V_f}\nabla.\left(F\vec{u}A_f\right)=-\frac{F}{V_f}\frac{\partial V_f}{\partial t} \quad (3)$$

V_f = volume fraction of fluid in each cell, ρ = density; $\vec{u}$ = velocity vector; A_f = fractional areas open to flow; P= pressure; t = viscous stress tensor, G = gravitational force; F is fluid fraction. It can be seen that, in cells completely full of fluid, V_f and A_f is equal to 1, thereby reducing the equations to the basic incompressible RANS equations.

A sub-model included in the commercial code is able to simulate the natural entrainment of air due to turbulence at the free surface. When any disturbance of size L_T at the free surface is associated with a larger energy per unit volume, P_T than the energy of the stabilizing forces, P_d the sub-model allows a volume of air to enter the mixture flow [11]. The equations of the sub-model are as follows:

$$L_T=c_\mu\left(\frac{3}{2}\right)^{1/2}\frac{k^{3/2}}{\varepsilon};\quad P_d=\rho_m g_n L_T+\frac{\sigma}{L_T};\quad P_T=\rho_m k \quad (4)$$

$$P_T>P_d:\delta V=C_{air}A_s\left[\frac{2(P_T-P_d)}{\rho_m}\right]^{1/2} \quad (5)$$

where g_n is the component of the vector of the acceleration of gravity in the direction normal to the free surface; σ is the surface tension; C_{air} is a coefficient of proportionality; As is the surface area; and δV is the volume of air allowed to enter the flow through the free surface per unit time. According to Hirt, a good first guess is C_{air} = 0.5, which assumes on average that air is trapped over about half the surface

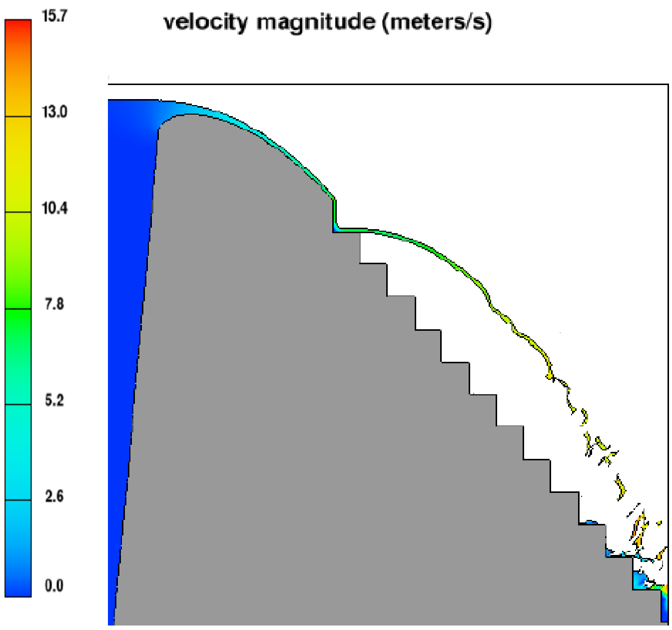
area of the raised disturbance [11]. In this numerical simulation the global tabs were specified with one fluid, incompressible flow and free surface or sharp interface. Also, the fluid properties were specified as those for water at 20 degrees Celsius for all simulations. Used geometry in this simulation was drawn in AutoCad Software. Surface roughness value was the inclusion of a typical concrete roughness (1 mm) value applied on the surface of all spillway geometry. To simulate a given flow, it is important that the boundary conditions accurately represent what is physically occurring. There are six different boundaries on the mesh to be fixed, plus the obstacle surface. The boundaries on the mesh and their coordinate directions were set as Table 1.
Boundary condition for the Top-Z direction was labeled as “symmetry”, which implies that identical flows occur on the other side of the boundary and hence there is no drag. In the Left-X and Right-X directions, “hydrostatic pressure “and “outflow” boundary conditions were used respectively. The “wall” function was applied in the Sides-y directions which involve null velocities normal to the spillway side walls. Determining the appropriate mesh domain along with a suitable mesh cell size is a critical part of any numerical model simulation. A sensitivity analysis was performed on the mesh cell size and the 20 cm cell size had best performance on hydraulic results and also time consumption in solving equations. In an effort to decrease the computational time required for a simulation to reach steady-state, constant water levels in the reservoir was used as the initial condition. Flow analysis was carried out for a time when a steady state was reached. This was determined by inspecting results such as the kinetic energy and the turbulence kinetic energy of the system.

Table 1. Applied boundary conditions

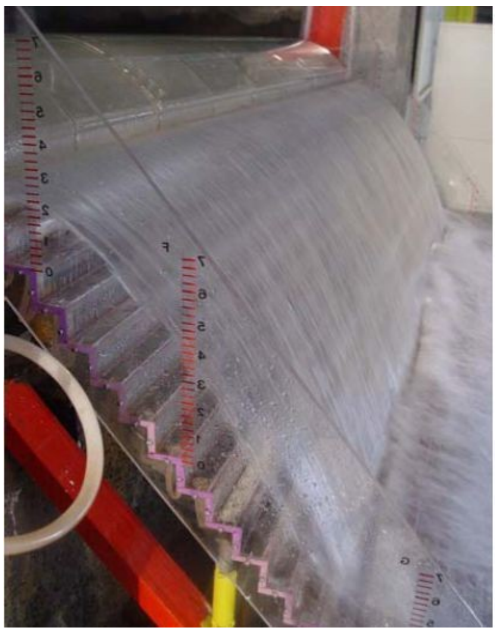
Boundary	Left-X	Right-X	Bottom-Z	Top-Z	Sides-y
Condition	hydrostatic pressure	outflow	wall	symmetry	wall

4. Results and discussions

At first, results of the original design of stepped spillway are presented.
Figure 3 shows the inflow condition of the original model, with the low discharge equal to 40m^3/s. In this condition the flow ignores the first step, and leaves it horizontally and passes other initial steps. As can be seen in Figure 3a, the numerical model is capable to show the free surface of flow and its jump in the low discharge as well as physical model (Figure 3b). As mentioned before to solve this undesirable flow condition, five alternatives (Figure 4) were tested and flow pattern were visualized. In these alternatives two first steps divided to different sizes.



(a)



(b)

Figure 3. Flow pattern in original design (Numerical and physical)

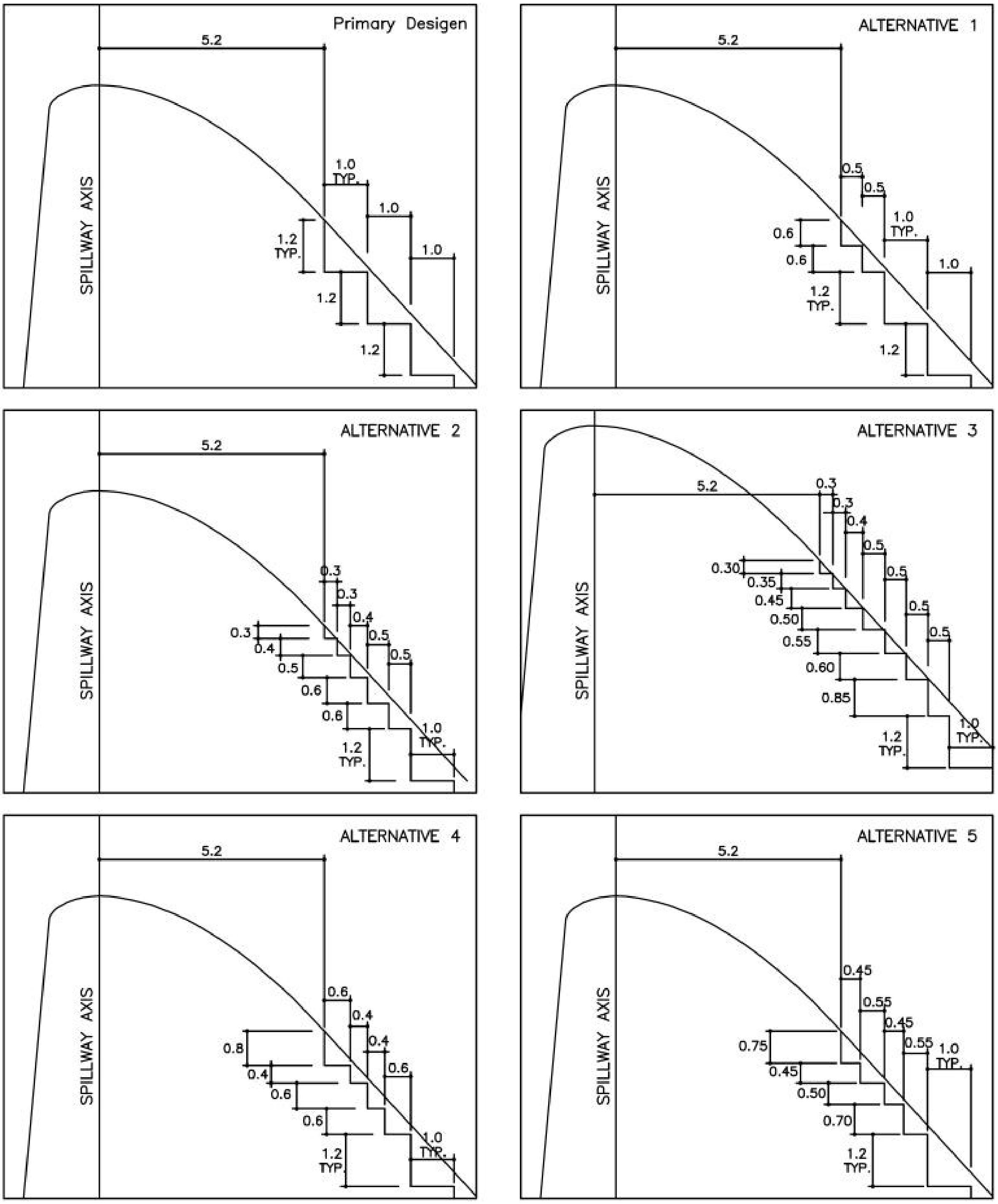


Figure 4. Suggested alternatives

Alternative 1 to 5 had positive effect on flow pattern; however, they could not properly improve the flow condition over the steps. For example, performance of the Alternative 1 is shown in Figure 5. In this Figure the half part of spillway model is the primary design scheme and another half is the new suggested alternatives which in this figure is Alternative 1. Cedex profile was also examined in the model and its performance was acceptable in the physical model and selected as the best alternative (Figure 6).

The numerical model was configured with H_d = 4.1m (for design flow rate = 1000 m^3/s) and steps geometry based on H_d and Cedex profile. Figure 7 illustrates result of the numerical and physical model for the same discharge equal to $40 m^3/s$. As can be seen, flow patterns in both experimental and numerical models were improved and flow passes over the spillway normally and jumping was removed completely.

Figure 5. Flow pattern in Alternative 1

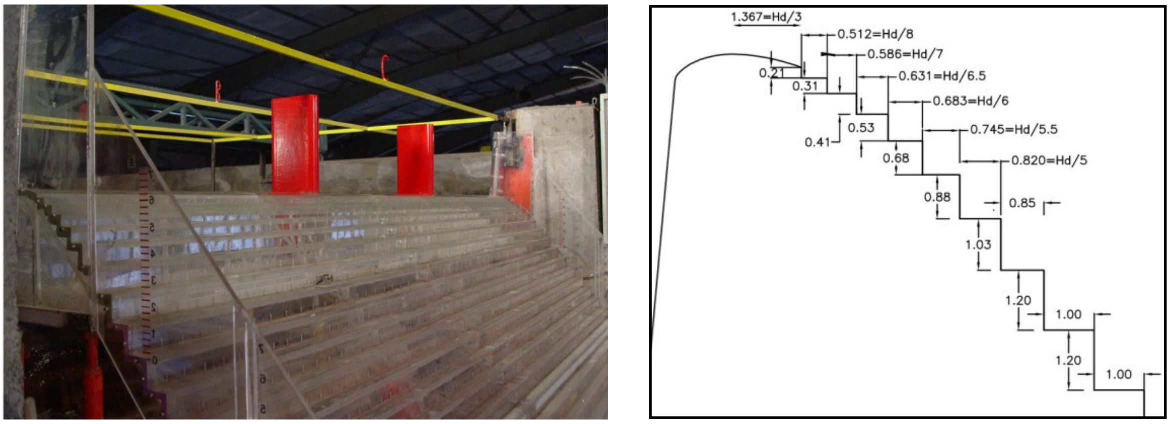


Figure 6. Cedex profile

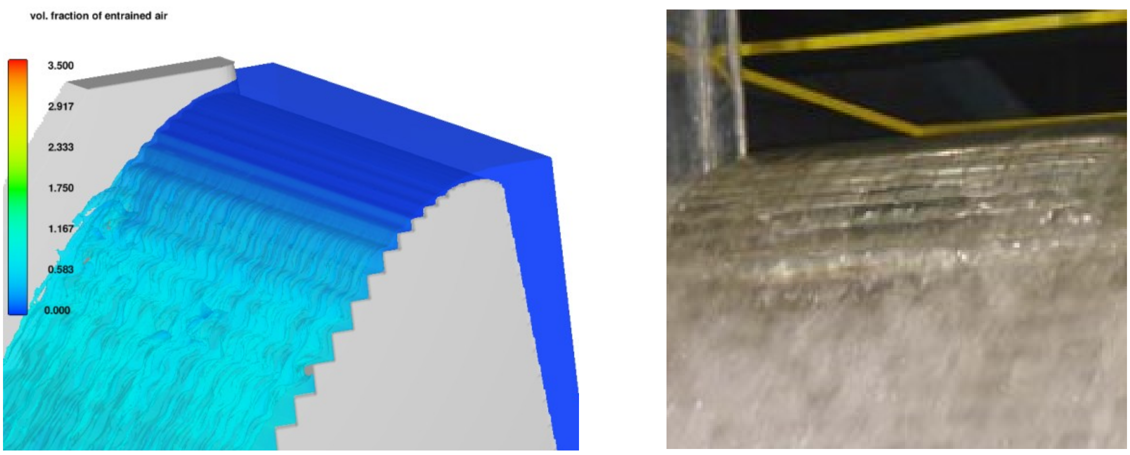


Figure 7. Flow pattern in Cedex profile (Numerical and physical)

To have a comparison between physical and numerical modeling results, characteristics of flow over stepped spillway in two different discharges 1000 m^3/s and 1600 m^3/s are presented in this section. Flow regime in this study was skimming over stepped spillway. Figures 8 and 9 show aeration over steps.

As can be seen in Figures 8 and 9, for lower discharge equal to 1000 m^3/s, large quantities of air entrain, upstream of the spillway. For higher discharge (1600 m^3/s) non-aerated region dominates large portions of the flow in the spillway. Figure 10 demonstrates velocity vectors together with pressure in step niches within the numerical modeling results for two discharges.

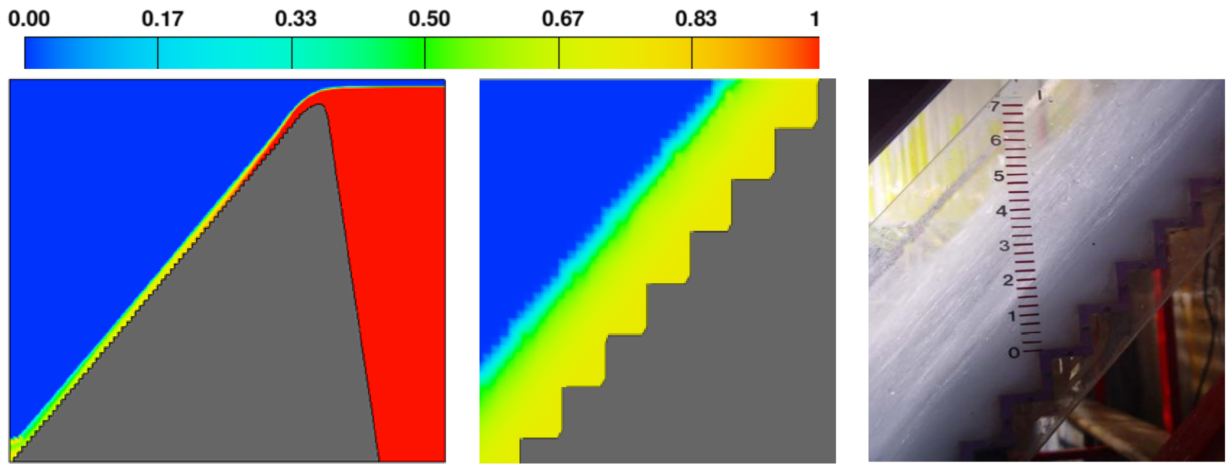


Figure 8. Flow aeration over steps (Q=1000 m^3/s)

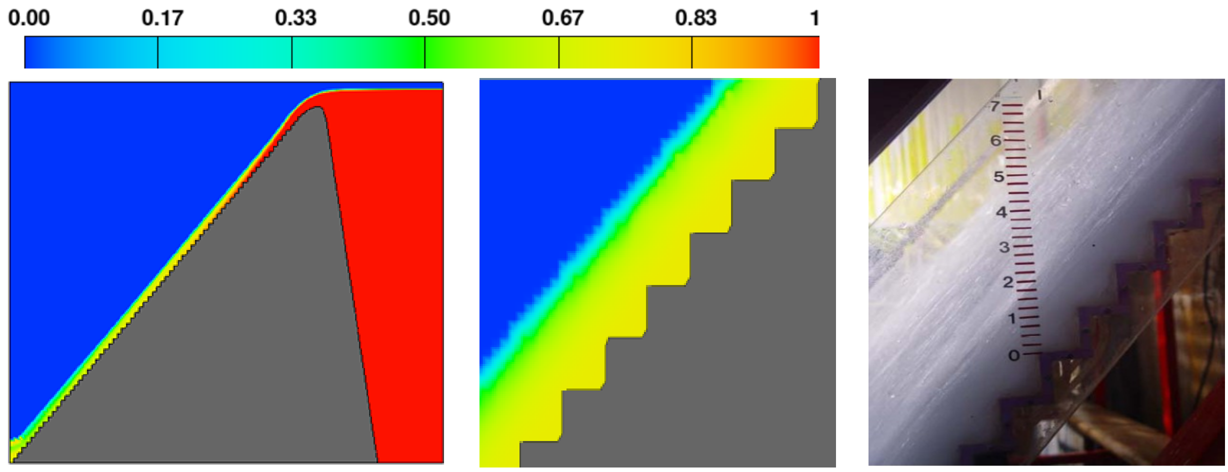


Figure 9. Flow aeration over steps (Q=1600 m^3/s)

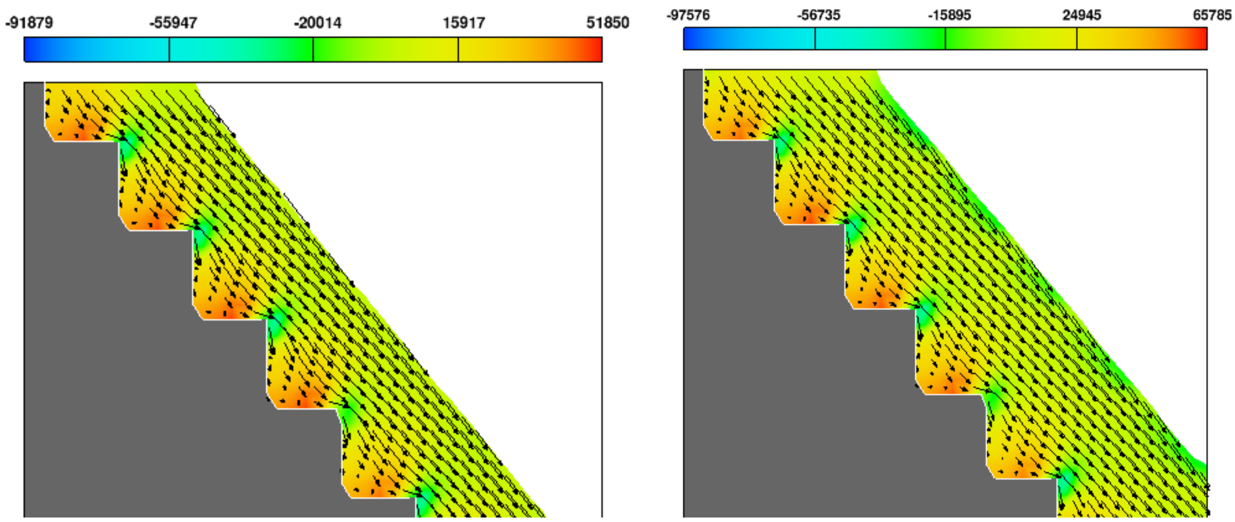


Figure 10. Velocity vectors together with pressure distribution (Q = 1000 and 1600 m^3/s)

Figure 10 indicates that the minimum values of pressure exist in the outer edge of the steps, close to the vertical walls of steps. This is caused by flow separation in this region when flow leaves the step which is clearly shown in the Figure 10 by the velocity vectors going out of the step edge. Also maximum pressure is located in the horizontal walls of the step near the edge, caused by the impact of the flow coming from the upper step. To determine the energy dissipation from upstream to downstream, results of the experiments with two flow rates were used. By using measured hydraulic characteristics of flow along the upstream and downstream of the physical model and based on the Bernoulli equation, total head loss in each case was calculated. Percent of dissipated energy in each case was then determined and plotted (Figures 11 and 12).

As can be seen from Figures 11 and 12, generally, the energy dissipation increased with increasing dimensionless horizontal distance from the spillway crest. In higher discharge the result of the simulation is closer to the physical model data.

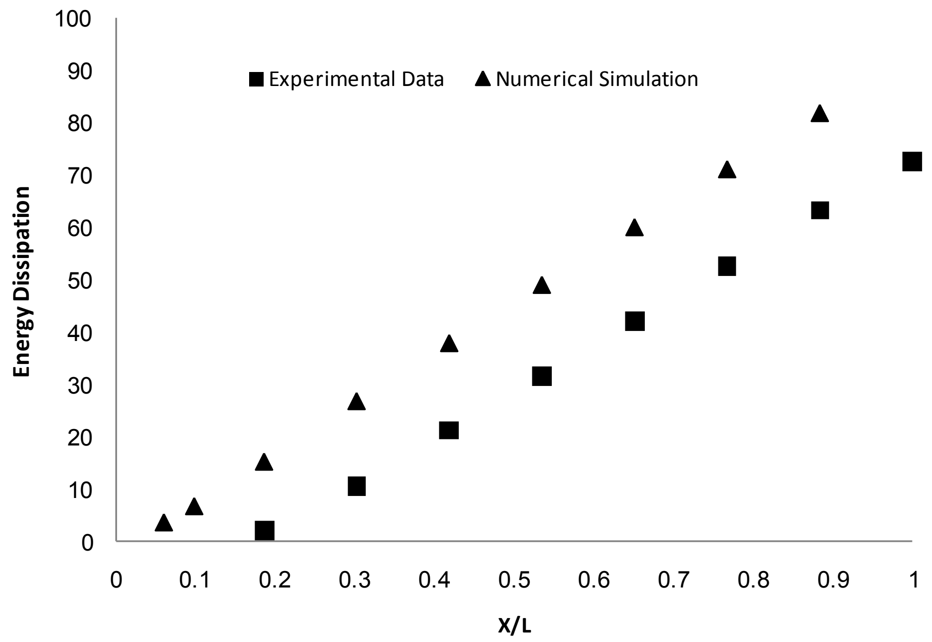


Figure 11. Energy dissipation, % in $Q = 1000$ m^3/s

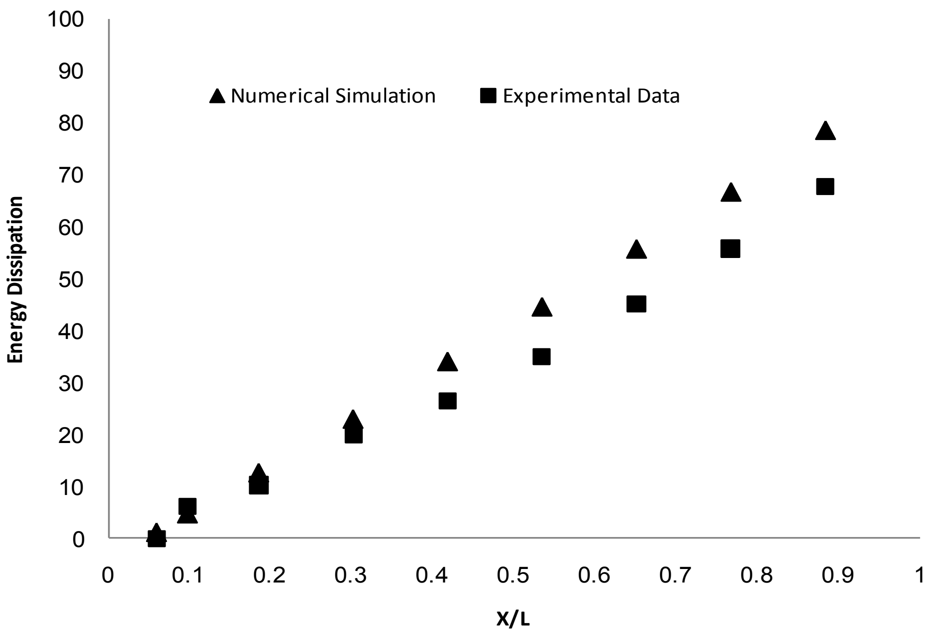


Figure 12. Energy dissipation, % in $Q = 1600$ m^3/s

5. Conclusions

This paper intends to present a combined numerical and experimental model of a stepped spillway. Joining the ogee spillway crest and the stepped chute profile is very important subject in design of stepped spillways. Regarding experimental and numerical simulations, the Cedex profile was selected as the best alternative. Also flow patterns predicted by numerical simulations and compared with the physical modeling observations. Flow over the steppes and energy dissipation along the spillway were computed numerically and compared with experimental data. Numerical and experimental results showed that the agreement was good. This points out that a proper numerical modeling of the proposed design before construction the physical model will result in cost and time saving of the project.

Acknowledgement

The Authors would like to thank from Water Research Institute (WRI), Tehran, Iran for their kindly cooperation in using physical model data.

References

[1] Minor H E, Hager W H. Hydraulics of Stepped Spillways. A. A. Balkema, Rotterdam, 2000, pp. 201.
[2] Chanson H. The Hydraulics of Stepped Chutes and Spillways. A. A. Balkema, Lisse, 2001, pp. 375.
[3] Song C C S, Zhou F. Simulation of Free Surface Flow over Spillway. J. Hydr. Engrg., 1999, 125(9). 959-967.
[4] Savage B M, Johnson M C. Flow over Ogee Spillway: Physical and Numerical Model Case Study. J. Hydr. Engrg., 2001, 127: 640-649.
[5] Ho D, Boyes K, Donohoo S, Cooper B. Numerical Flow Analysis for Spillways. 43rd ANCOLD Conf., Hobart, Tasmania, 2003, pp. 24-29.
[6] Gessler D. CFD Modeling of Spillway Performance. Proc. World Water and Environmental Resources Cong., Anchorage, Alaska. 2005, 15-19.
[7] Bombardelli F A, Meireles I, Matos J. Laboratory measurements and multi-block numerical simulations of the mean flow and turbulence in the non-aerated skimming flow region of steep stepped spillways. J. Environmental Fluid Mechanics, 2011, Vol. 11, Issue 3, pp. 263-288.
[8] Chamani M R. Air inception in skimming flow regime over stepped spillways. A.A. Balkema, Rotterdam NL, 2000, pp. 61-67.
[9] Pfister M, Hager W H. Self-entrainment of air on stepped spillways. Int. J. Multiphase Flow. Volume 37, Issue 2, 2011, pp. 99-107.
[10] Chen, Q., Dai, G. and Liu, H. Volume of fluid model for turbulent numerical simulation of stepped spillway overflow. J. Hydr. Engrg., Vol. 128, No. 7, 2002. pp. 683-688.
[11] Hirt CW. Modeling turbulent entrainment of air at a free surface. Technical Note 61, Flow Science, 2003, Inc (FSi-03-TN61).

Hamed Sarkardeh receivedhis PhD in Civil Engineering from the Iran University of Science and Technology Tehran, his MSc from the Amirkabir University of Technology (Tehran Polytechnic) and his BSc degree from the Ferdowsi University of Mashhad in 2013, 2009 and 2006,respectively. He also completed a post-doc in Mechanical Engineering at the Amirkabir University of Technology (Tehran Polytechnic). He is currently an Assistant Professor at Department of Civil Engineering of the Hakim Sabzevari University whose main research interests are Fluid Mechanics, Hydropower Plants, Dam Engineering, Energy and Sustainability with a focus on renewable energy.
Email address: sarkardeh@hsu.ac.ir

Morteza Marosi received his MSc degree in Hydraulic Structures from the Shahid Chamran University, Ahvaz and BSc from the Ferdowsi University of Mashhad, Iran, in 2009 and 2007 respectively. He is currently one of the senior researchers in Hydraulic Structures Division of Water Research Institute (WRI) in Iran. His research interests include physical and numerical modeling of hydraulic structures, hydropower dams and turbidity current in dam reservoirs.
E-mail address: m.marousi@wri.ac.ir

Raza Roshan was graduated MSc and BSc in Hydraulic Structures from University of Tehran, Tehran, Iran in 1996. Currently, he is head of the Hydraulic Structures Division of Water Research Institute (WRI). He is also senior member of Iranian hydraulic association and has published more than 50 papers in national and international journals and conference.
Email address: r.roshan@wri.ac.ir