6th National Congress on Civil Engineering,
April 26-27, 2011, Semnan University, Semnan, Iran

Numerical Simulation of Uniform Flow Region over a Steeply Sloping Stepped Spillway

Mohammad Sarfaraz[1], Jalal Attari[2]
1- Graduate student, Department of Civil Engineering, Sharif University of Technology, Tehran
2- Assistant professor, Water Engineering Department, Power & Water University of Technology, Tehran

mohammad.sarfaraz@gmail.com

Abstract

Stepped spillways have gained much interest in recent decades because of their compatibility with Roller Compacted Concrete (RCC) dams. Hydraulics of stepped spillways is not simple considering different flow regimes and regions along the chute. Estimation of flow characteristics on the stepped chutes is presently carried out by using some empirical formulae and physical modeling. However, this can be improved by application of Computational Fluid Dynamics (CFD) models. In this paper, flow characteristics within the uniform region of Javeh stepped spillway, located on the body of a RCC dam, was computed by a commercial CFD program. A comparison of the numerical and physical model results showed a relatively good agreement. The study indicates that the turbulence numerical simulation is an effective and useful method for the complex stepped spillway overflow.
Keywords: Hydraulic Structures, Stepped Spillways, Flow Characteristics, Numerical modelling, FLOW-3D

1. INTRODUCTION

Stepped spillways are found to be effective for energy dissipation of excess flood released from dams (e.g. Monksville & Upper Stillwater) [1,2]. Many studies [1,3,4] have shown that favorable design of stepped spillways can decrease the size of stilling basins significantly and thus saving on construction costs. Stepped spillways have gained much interest in recent decades because of their compatibility with Roller Compacted Concrete (RCC) dams [1]. Once a stepped chute is located on the body of a RCC dam, it offers additional constructional and economical advantages.

The flow over a stepped spillway can be classified into three types: nappe flow, transition flow and skimming flow [5]. On stepped chutes with skimming flow regime, the flow is highly turbulent. Once the outer edge of boundary layer reaches free surface, natural air entrainment commences (Figure1). Beyond this inception point, an air water mixture layer forms which gradually extends through the flow. Far downstream, flow will become quasi-uniform in a long chute and the depth will not vary at this equilibrium condition for a given flow [6]. L_u designates distance, along the chute, between the ogee crest and the section where quasi–uniform flow forms.

For skimming flow over the stepped spillway, energy dissipation occurs due to: i) recirculation between the main flow and the water trapped on the steps, ii) continuous production of large vortices and their break off and transport into the skimming stream [6]. If the flow reaches to uniform/quasi uniform flow condition, its characteristics can be readily used for estimation of energy loss over the stepped chute. On this basis a stilling basin is usually designed at the toe of the spillway for dissipation of the residual energy.

According to the conventional code of practice, designers calculate flow characteristics (i.e. velocity, depth and air concentration) over the stepped spillways by selecting some relationships among a variety of empirical formulae and design charts. These initial designs are further tested by physical models which are valuable but expensive and time consuming [3,4]. Nowadays, with the availability of high-performance computers and commercial CFD codes, flow characteristics over hydraulic structures can be quickly estimated by these numerical models which are highly needed for initial design purposes.

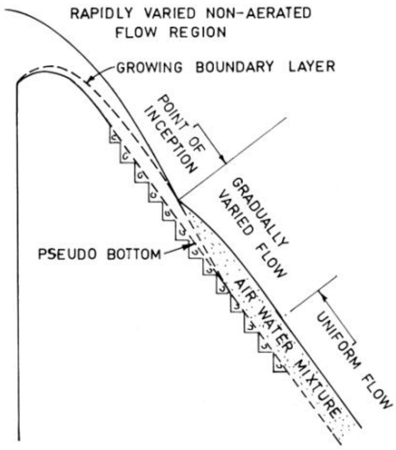

Figure 1. Flow regions in skimming flow (after [6])

Javeh is an RCC dam, currently under construction near Sanandadj. A schematic of the Javeh stepped spillway is shown in Figure 2. Width of the spillway and design discharge are 55m and 970m^3/s respectively. Below the tangency point, the spillway profile has a slope of 1.2V:1.0H. On the entire spillway, there were 70 steps with the height of 1.2 m and length of 1.0 m.

In this paper, a commercial CFD code (FLOW-3D) was employed to estimate flow characteristics over the Javeh stepped spillway. Particular attention was given to the uniform flow region over the stepped chute. Velocity magnitudes and flow depths within this region can be estimated to investigate efficiency of energy dissipation.

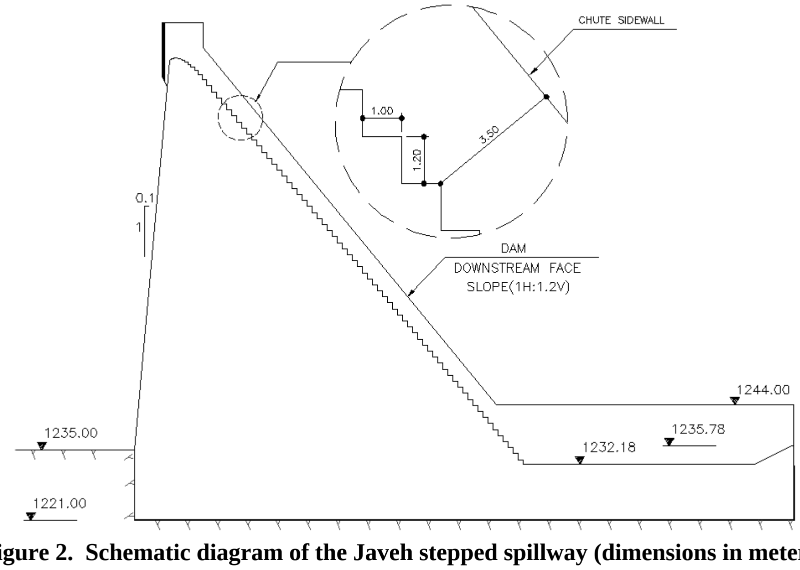

Figure 2. Schematic diagram of the Javeh stepped spillway (dimensions in meters)

2. PHYSICAL MODEL

Physical model testing of Javeh spillway, with a scale of 1:25, was recently carried out in the hydraulics laboratory of Water Research Institute (WRI) in Iran.

Flow parameters were measured for seven flow rates (Q). The flow velocities and depths were measured by a pitot tube with single point measurement at 0.6 flow depth and a ruler installed on the wall of the chute respectively. Critical depths (d_c) and their ratios to the step height (d_c/h) were calculated for the range of experiments' discharges (Table 1).

Table 1- Flow parameters for Javeh spillway (scaled up to the prototype)

Q (m^3/s)	d_c (m)	d_c/h
200	1.10	0.92
400	1.75	1.46
500	2.03	1.69
800	2.78	2.32
970	3.17	2.64
1600	4.42	3.68
2100	5.30	4.42

3. NUMERICAL MODEL

3.1. Numerical Concepts

In this part, governing equations which have special features due to air-water flow in the stepped spillways are presented.

3.1.1. Mass Continuity Equation

The general mass continuity equation is:

$$V_F \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}\left(\rho u A_x\right) + R\frac{\partial}{\partial y}\left(\rho v A_y\right) + \frac{\partial}{\partial z}\left(\rho w A_z\right) + \xi \frac{\rho u A_x}{x} = R_{DIF} + R_{SOR} \tag{1}$$

Where V_F is the fractional volume open to flow, ρ is the fluid density, R_{DIFF} is a turbulent diffusion term and R_{SOR} is a mass source. A_x is the fractional area open to flow in the x-direction, A_y and A_z are similar area fractions for flow in the y and z directions, respectively. The coefficients R and ξ depend on the choice of the coordinate system, which $R = 1$ and $\xi = 0$ in the Cartesian coordinate [7]. The first term on the right side of Equation (1), is a turbulent diffusion term,

$$R_{DIF} = \frac{\partial}{\partial x}\left(\upsilon_\rho A_x \frac{\partial \rho}{\partial x}\right) + R\frac{\partial}{\partial y}\left(\upsilon_\rho A_y \frac{\partial \rho}{\partial y}\right) + \frac{\partial}{\partial z}\left(\upsilon_\rho A_z \frac{\partial \rho}{\partial z}\right) + \xi \frac{\rho \upsilon_\rho A_x}{x} \tag{2}$$

Where the coefficient υ_ρ is equal to $cp\mu/\rho$, in which μ is the coefficient of momentum diffusion (i.e., the viscosity) and c_p is a constant whose reciprocal is usually referred to as the turbulent Schmidt number [8].

3.1.2. Momentum Equations

The equations of motion in the three coordinate directions are the Navier-Stokes equations with some additional terms:

$$\begin{aligned}
&\frac{\partial u}{\partial t} + \frac{1}{V_F}\left\{uA_x \frac{\partial u}{\partial x} + vA_y R\frac{\partial u}{\partial y} + wA_z \frac{\partial u}{\partial z}\right\} - \xi \frac{A_y v^2}{xV_F} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + G_x + f_x - b_x - \frac{R_{SOR}}{\rho V_F}\left(u - u_w - \delta u_s\right) \\
&\frac{\partial u}{\partial t} + \frac{1}{V_F}\left\{uA_x \frac{\partial v}{\partial x} + vA_y R\frac{\partial v}{\partial y} + wA_z \frac{\partial v}{\partial z}\right\} + \xi \frac{A_y uv}{xV_F} = -\frac{1}{\rho}\left(R\frac{\partial p}{\partial y}\right) + G_y + f_y - b_y - \frac{R_{SOR}}{\rho V_F}\left(v - v_w - \delta v_s\right) \\
&\frac{\partial w}{\partial t} + \frac{1}{V_F}\left\{uA_x \frac{\partial w}{\partial x} + vA_y R\frac{\partial w}{\partial y} + wA_z \frac{\partial w}{\partial z}\right\} = -\frac{1}{\rho}\frac{\partial p}{\partial z} + G_z + f_z - b_z - \frac{R_{SOR}}{\rho V_F}\left(w - w_w - \delta w_s\right)
\end{aligned} \tag{3}$$

In these equations, (G_x,G_y,G_z) are body accelerations, (f_x,f_y,f_z) are viscous accelerations, (b_x,b_y,b_z) are flow losses in porous media or across porous baffle plates and the final terms account for the injection of mass at a source represented by a geometry component [8].

3.1.3. VOF Method

Several methods have been used to approximate free surface flows. A simple, but powerful method is the VOF method. This method is shown to be more flexible and efficient than other methods for treating complicated free surface flows. It was designed for two or more immiscible fluids where the position of the interface between the fluids is of interest. The use of several points in a cell to define the region occupied by a certain fluid therefore, seems unnecessarily excessive [7,10].

Fluid configurations are defined in terms of a volume of fluid function, F(x,y,z,t). This function represents the volume of fluid #1 per unit volume and satisfies the equation:

$$\frac{\partial F}{\partial t}+\frac{1}{V_F}\left\{\frac{\partial}{\partial x}\left(FA_x u\right)+R\frac{\partial}{\partial y}\left(FA_y v\right)+\frac{\partial}{\partial z}\left(FA_z w\right)+\xi\frac{FA_x u}{x}\right\}=F_{DIF}+F_{SOR} \quad (4)$$

where

$$F_{DIF}=\frac{1}{V_F}\left\{\frac{\partial}{\partial x}\left(\upsilon_F A_x\frac{\partial F}{\partial x}\right)+R\frac{\partial}{\partial x}\left(\upsilon_F A_y R\frac{\partial F}{\partial y}\right)+\frac{\partial}{\partial z}\left(\upsilon_F A_z\frac{\partial F}{\partial z}\right)+\xi\frac{\upsilon_F A_x F}{x}\right\} \quad (5)$$

The diffusion coefficient is defined as $\upsilon_F = cF\,\mu/\rho$ where cF is a constant whose reciprocal is sometimes referred to as a turbulent Schmidt number. This diffusion term only makes sense for the turbulent mixing of two fluids whose distribution is defined by the F function.

3.1.4. RNG k-ε Turbulent Model

The RNG turbulence model solves for turbulent kinetic energy (k) end turbulent kinetic energy dissipation rate (ε). The RNG-based models rely less on empirical constants while setting a framework for the derivation of a range of parameters to be used at different turbulence scales [9].

The RNG-based k-ε turbulence model is derived from the instantaneous Navier-Stokes equations, using a mathematical technique called “renormalization group” (RNG) method. The k-ε equations are as follows:

$$\frac{\partial\varepsilon}{\partial t}+\frac{1}{V_F}\left\{uA_x\frac{\partial\varepsilon}{\partial x}+vA_y R\frac{\partial\varepsilon}{\partial y}+wA_z\frac{\partial\varepsilon}{\partial z}\right\}=\frac{CDIS1.\varepsilon}{k}\left(P_T+CDIS3.G\right)+Diff_\varepsilon-CDIS2.\frac{\varepsilon^2}{k} \quad (6)$$

Here CDIS1, CDIS2 and CDIS3 are all dimensionless parameters. CDIS1 has the default value of 1.42, but CDIS2 and CDIS3 are computed from the turbulent kinetic energy (k) and turbulent production (P_T). The diffusion of dissipation is:

$$Diff_\varepsilon=\frac{1}{V_F}\left\{\frac{\partial}{\partial x}\left(\upsilon_\varepsilon A_x\frac{\partial\varepsilon}{\partial x}\right)+R\frac{\partial}{\partial y}\left(\upsilon_\varepsilon A_y R\frac{\partial\varepsilon}{\partial y}\right)+\frac{\partial}{\partial z}\left(\upsilon_\varepsilon A_z\frac{\partial\varepsilon}{\partial z}\right)+\xi\frac{\upsilon_\varepsilon A_x\varepsilon}{x}\right\} \quad (7)$$

3.1.5. Air Entrainment Sub-model

In stepped spillways overflow, the turbulence is sufficient to disturb the surface to the point of entraining air into the flow [5]. Turbulence transport models characterize turbulence by a specific turbulent kinetic energy, k and a dissipation function ε. A characteristic size of turbulence eddies is then given by [8]:

$$L_t=c_\mu\left(\frac{3}{2}\right)^{0.5}\frac{k^{1.5}}{\varepsilon} \quad (8)$$

The disturbance kinetic energy per unit volume associated with a fluid element raised to a height L_t and with surface tension energy based on a curvature of L_t, is:

$$P_d=\rho g_n L_t+\frac{\sigma}{L_t} \quad (9)$$

Where ρ is the liquid density, σ its coefficient of surface tension and g_n is the component of gravity normal to the free surface.

For air entrainment to occur the turbulent kinetic energy per unit volume, (P_t=ρk) must be larger than P_d, i.e., the turbulent disturbances must be large enough to overcome the surface stabilizing forces. The volume of air entrained per unit time, δV, should be proportional to the surface area, A_s, and the height of the disturbances above the mean surface level. All together the following equation could be written:

$$\delta V=C_{air}A_s\left(\frac{2\left(P_t-P_d\right)}{\rho}\right)^{0.5} \quad (10)$$

where C_{air} is a coefficient of proportionality. A good first guess is C_{air}=0.5, i.e., assume on average that air will be trapped over about half the surface area [11].

3.1.6. Drift Flux Sub-model

Flow over stepped spillways is composed of multiple components (i.e. fluid/bubbles, fluid/fluid mixtures), where the components have different densities, it is observed that the components can assume different flow velocities [4]. Velocity differences arise because the density differences result in non-uniform body forces. The goal of the drift-flux model is to compute the motion of the two phases relative to the volume-averaged velocity, $\bar{u}$ [7]. The volume-weighted average velocity is:

$$\bar{u} = f u_1 + (1-f) u_2 \tag{11}$$

For two phase in stepped spillways, the following equation could be derived:

$$\frac{\partial u_r}{\partial t} + u_2.\nabla u_2 - u_1.\nabla u_1 = \left(\frac{1}{\rho_1} - \frac{1}{\rho_2}\right)\nabla P - \left(\frac{1}{f\,\rho_1} - \frac{1}{(1-f)\,\rho_2}\right) k_d u_r \tag{12}$$

Where k_d is a drag coefficient that relates the interaction of the two phases, and u_r is the relative velocity difference between the dispersed and continuous phases. The goal is to determine the relative velocity u_r.

3.2. Numerical Procedure

A commercially available Computational Fluid Dynamics program (Flow-3D) was used for solving the Reynolds-averaged Navier-Stokes equations in combination with the RNG K-ε eddy-viscosity closure model. The software solves the fully three dimensional transient Navier-Stokes equations using the FAVOR[1] and VOF method [10]. The solver uses finite volume approximation to discretize the computational domain. The pressure and velocity are coupled implicitly by using the time-advanced pressures in the momentum equations and time-advanced velocities in the continuity equations. It solves these semi-implicit equations iteratively using relaxation techniques. FAVOR defines solid boundaries and determines fractions of areas and volumes (open to flow) in partially blocked volume to compute flows correspondent to those boundaries. In this way, boundaries and obstacles are defined independently of grid generation, avoiding saw tooth representation of the use of body-fitted grids. The RNG model was selected for the current simulation based on the recommendations of [11,12,13].

3.2.1. Model Geometry and Computational Grid

The model geometry of the stepped spillway was generated by AutoCAD in prototype dimensions. The domain was discretized using one non-uniform mesh block. The evolution in time was used as a relaxation to the final steady state. The steady state was checked through monitoring the flow kinetic energy.

3.2.2. Boundary Conditions

Boundary condition for the 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 x direction the boundary condition was “specified stagnation pressure”. With this algorithm, Flow-3D is able to model various flow heights beginning at a stagnation pressure state. The “wall function” was applied in the y direction which involves null velocities normal to the spillway side wall. It is important to note that these fluid heights coincided with the initial conditions selected for the coarse grid to perform simulations in the most time-efficient manner. The computational domain including boundary conditions is shown in Figure 3.

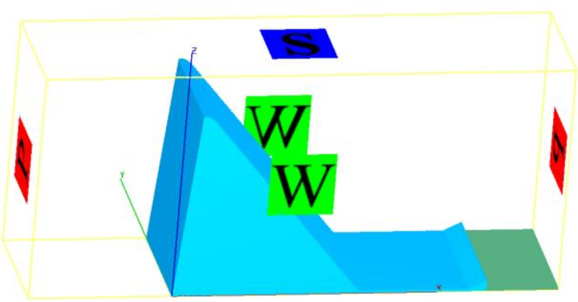

Figure 3. Configurations of boundary conditions

[1] Fractional Area/Volume Obstacle Representation

3.2.3. Initial Conditions

In an effort to decrease the computational time required for a simulation to reach steady-state, simulations were first carried out on a coarse mesh. Results of this rough simulation were then used as input data for simulation of the same situation but with a finer mesh. This approach, introduced by [14], was employed to damp effects of a wave that is caused by a sudden motion of the fluid.
The wave propagates along the channel length and reflects back from the upstream and downstream boundaries of the domain until it is eventually dissipated by the viscous forces of the fluid. Constant water levels in the reservoir and tailwater elevations in the stilling basin were used as the upstream and downstream initial conditions respectively.

4. NUMERICAL RESULTS AND ANALYSIS

The experimental data by Sarfaraz [4] were used for validation of flow simulation over the stepped spillway. The comparison includes the velocity, flow depth and location of uniform flow region towards the end of the stepped chute.

4.1. Velocity Distribution

Figure 4 shows the calculated velocity contours along the stepped spillway for the discharge of 970m^3/s. Actually, the flow velocities increase in the upper parts of the stepped spillway but tend to a constant value within the uniform flow region towards the end of the chute. Velocity distribution across the vertical direction, magnified in Figure 4, shows that the velocity magnitudes are increasing from the bottom of the steps towards the flow surface.

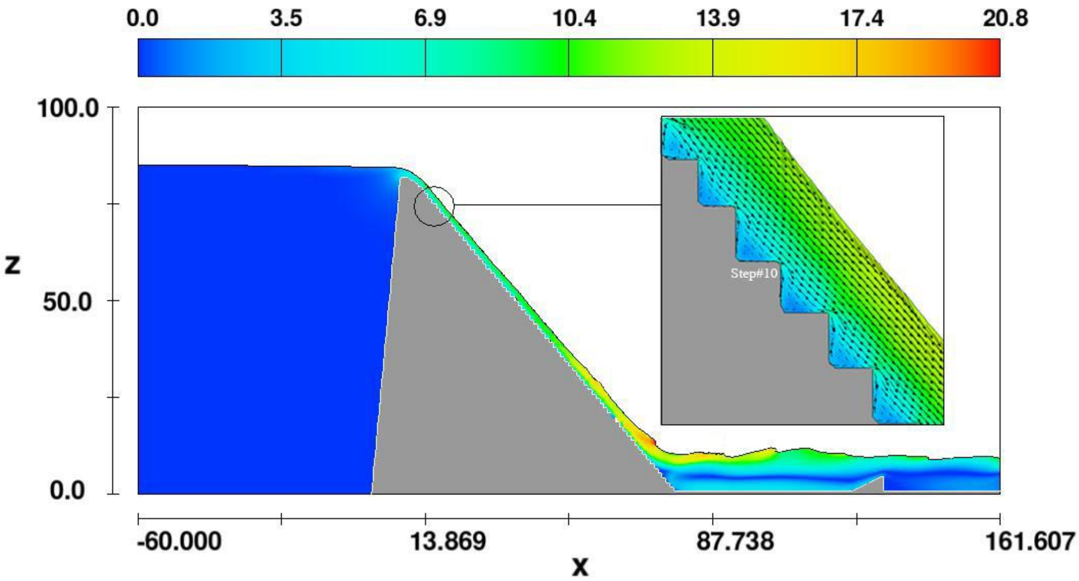

Figure 4. Velocity contours and vectors along the stepped spillway, Q= 970 m^3/s

A comparison of computed and measured mean velocity magnitudes, within the flow uniform region, for different flow rates reveals a good agreement (Figure 5).

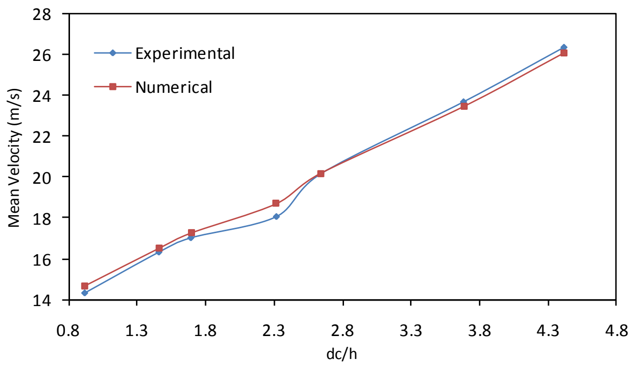

Figure 5. Comparison of computed and measured mean velocities in the uniform flow region

4.2. Fluid Depths

Figure 6 demonstrates flow depths perpendicular to the pseudo-bottom within the uniform flow region. A good agreement can be observed between numerical and experimental results.

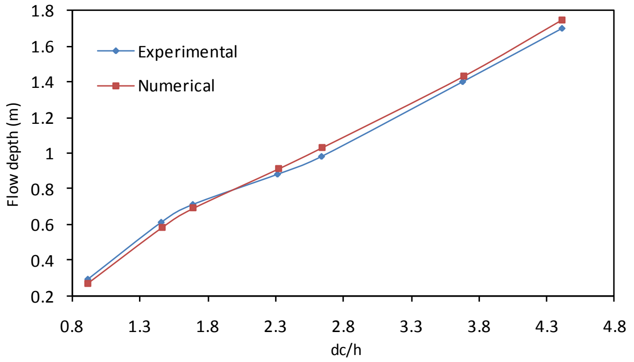

Figure 6. Comparison between computed and measured depths within the uniform flow region

4.3. Location of Uniform Flow Region

Figure 4 demonstrates calculated velocities along the stepped spillway for the discharge of 970m^3/s. From this Figure, the location of uniform flow region can be specified. L_u designates distance, along the chute, between the ogee crest and the section where quasi–uniform flow forms. Results of L_u for various discharges, expressed in terms of the ratio of critical depth to step height (d_c/h), are shown in Figure 7. Results reveals that by increasing flow rate, position of the uniform flow region shifts towards downstream of the chute. A comparison of the computed results with the experimental data shows a satisfactory agreement. The discrepancies for higher values of d_c/h could be attributed to the fact that distance between discrete measurement points at physical model were larger towards the end of the chute which reduces the accuracy of estimation of L_u on the basis of physical model results.

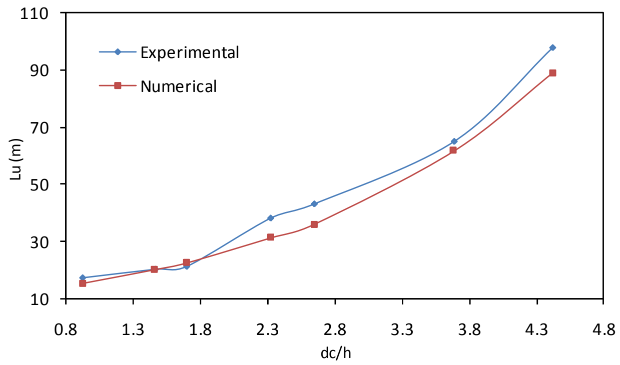

Figure 7. Comparison of computed and observed values of L_u

5. CONCLUSIONS

In this paper, a commercially available CFD code (FLOW-3D) was employed to investigate flow characteristics over Javeh stepped spillway, located on the body of a RCC dam. The computational results were compared with the experimental data in three respects: mean velocity, fluid depth and location of uniform flow region. On the basis of results, the following conclusions can be drawn:

- The RNG turbulence model with the VOF method and Drift Flux sub-model can simulate the stepped spillway overflow successfully. The VOF model of the air and water phases can well track the free surface based on the time-dependent simulation.
- By increasing the flow rate, location of the uniform flow region shifts towards downstream of the stepped chute.
- The velocity and water depth within the uniform flow region can be readily used to estimate the energy dissipation efficiency of the stepped spillway.
- Flow characteristics on the stepped spillways, at least within the uniform flow region, can be estimated with a good accuracy by CFD commercial codes.
- Considering advantages of the CFD models (e.g. less time and lower cost), flow simulation can be started by application of numerical models and subsequently be refined with physical model testing, if necessary.

6. ACKNOWLEDGMENT

The physical model tests of the Javeh stepped spillway were carried out in the Water Research Institute. The Client for this project was Iran Water and Power Development Company. The authors are grateful to them for permission to use un-published data of the study for comparison with results of the current research.

7. REFERENCES

1. USBR Hydraulic Laboratory, (2006), "*Research State-of-art and Needs for Hydraulic Design of Stepped Spillways"*, HL-2005-06, U.S.A.
2. USBR Hydraulic Laboratory, (1987), "*Hydraulic Model Study of Upper Stillwater Dam Stepped Spillway & Outlet works*", REC-ERC-87-6, U.S.A.
3. Sarfaraz, M., Attari, J., Roshan, R. and Khorasanizadeh, A., (2010), *"Comparison of Empirical Relationships for Energy Dissipation with Physical Model Data of Stepped Spillways in Iran"*, 9th Iran's Hydraulic Conference, Tehran, (in Persian).
4. Sarfaraz, M., (2010), *"Hydraulic Design and Modelling of Stepped Spillways Located on Body of RCC Dams, Case Study: Javeh Stepped Spillway"*, A thesis submitted in partial fulfillment of the requirement for the degree of Bachelor of Science, Power and Water University of Technology (Shahid Abbaspoor), Tehran, (in Persian).
5. Chanson, H., (2010), *"The Hydraulics of Stepped Chutes and Spillways"*, Balkema, Lisse.
6. Khatsuria, R.M., (2005), *"Hydraulics of Spillways and Energy Dissipators"*, Marcel Dekker, New York, U.S.A.
7. Yeoh, G.H. and Tu, J., (2010), *"Computational techniques for multi-phase flows"*, Elsevier Ltd., U.S.A., pp 407-500.
8. Szymkiewicz, R., (2010), *"Numerical Modeling in Open Channel Hydraulics"*, Springer, Poland, pp 263-300.
9. Yakhot, V. and Smith, L.M., (1992), *"The Renormalization Group, the E-expansion and Derivation of Turbulence Models"*, Journal of Scientific Computing, Vol.7, pp 35-61.
10. Flow Science, Incorporated, (2008), *"FLOW-3D Users Manual Version 9.3"*, Santa Fe, New Mexico.
11. Bombardelli, F.A., Meireles, I. and Matos, J., (2010), *"Laboratory measurement and multi-block numerical simulations of the mean flow and turbulence in the non-aerated skimming flow region of steep stepped spillways"*, Environ Fluid Mechanics.
12. Tabbara, M., Chatila, J. and Awwad R., (2005), *"Computational simulation of flow over stepped spillways"*, Journal of Computers and Structures, Vol.83.
13. Chen, Q., Dai, G. and Liu, H., (2002), *"Volume of Fluid Model for Turbulence Numerical Simulation of Stepped Spillway Overflow"*, Journal of Hydraulic Engineering, Vol.128 (7), pp 683-688.
14. Savage, B.M., Johnson, M.C., (2001), *"Flow over Ogee Spillway: Physical and Numerical Model Case Study"*, Journal of Hydraulic Engineering, Vol.126 (8), pp 640-649.