EVALUATION OF HYDRAULIC PARAMETERS OF STEPPED SPILLWAY BY NUMERICAL MODELING

Reze Mohammadpour[1], Mahdi Attar[2], Aminuddin Ab. Ghani [3] ,H. Md. Azamathulla[4]

[3]PhD student, REDAC, Universiti Sains Malaysia, Engineering Campus, Seri Ampangan, 14300 Nibong Tebal, Penang, Malaysia.

Email: reza564@gmail.com

[2] Dpartment of civil engineering , shiraz branch, azad university ,shiraz, iran.

[3]Proefessor and Deputy Director, REDAC, Universiti Sains Malaysia, Engineering Campus, Seri Ampangan, 14300 Nibong Tebal, Penang, Malaysia. Email: redac02@eng.usm.my

[4]Associate Professor, REDAC, Universiti Sains Malaysia, Engineering Campus, Seri Ampangan, 14300 Nibong Tebal, Penang, Malaysia.

Email: redacazamath@eng.usm.my

In the present study flow patterns over a stepped-spillway with standard WES profile and free-surface flow was studied using the numerical modeling. The Fluent was employed to solve the Reynolds-averaged Navier-Stokes equation to predict dissipation of energy on the stepped spillway. To simulate the free-surface flow and turbulence two methods of Volume of Fluid (VOF) and K- ε standard were applied respectively. Both structured and unstructured meshes were used for simulation of spillway. To obtain more accurate result a dense mesh was used near steps, walls and also at free surface flow. The hydraulic parameters of step spillway such as velocity coefficient "φ", damping ration "K" and resistance coefficient "ζ" were selected to validate the model. A statistical comparison between experimental data and simulation showed the model can predict stepped spillway overflow with high accuracy. Due to less time demand and lower cost of the numerical method than that of experiments, numerical method has significant advantage in practical projects.

Keywords: energy dissipation; VOF; numerical modeling; stepped Spillway; K- ε

Introduction

The stepped spillways meaningfully decrease dimension of stilling basin in front of the dams and lead to have a large economic benefit.. The profile of stepped spillway is similar to same Ogee spillway profile. The energy of flow is dissipated in process of passing over the steps. Recently, Computational Fluid Dynamics (CFD) was used in different range of water resource and hydraulic structure due to less time and lower cost of the numerical modelling. Boes et al. (2003) investigated stepped spillway in large model flume with fiber-optical instrumentation. Bhajantri et al. (2007) have studied flow over spillway with gate operation by using numerical model. The simulation was developed using the finite volume method based on weakly compressible flow equations and large Eddy simulation (LES)-based turbulence model. Olsen et al. (1998) simulated the flow over a spillway in two and tree dimension. The model solved the Navier-Stokes equations using the k-ε turbulence method on a structured non-orthogonal grid. Johnson et al. (2006) compared numerical and physical data over ogee spillway in the presence of tailwater. The comparison showed that numerical modeling can accurately predict the rate of flow over the spillway and the pressure distribution on the spillway. The main objective of this research is simulation of stepped spillway with standard WES profile and similar step, to determined the hydraulic parameters/characteristics of stepped spillway such as velocity coefficient "φ" , damping ration "K" and resistance coefficient "ζ".

Experimental data

To validate the results of numerical modelling, a series of experimental data was collected from studies by Yang et al. (2001). Their model had a tank in the upstream with a spillway immediately after it. The high (P1)and width spillway was equal to 1.22 m and 0.3m respectively. A horizontal floor was jointed to steps by a curved bucket with 0.3 m radius. As shown in Fig.1, the spillway slope was equal to 3H:4V which connected from crest to toe by 18 steps. The discharge rates was changed between 4L/s to 60 L/s. General layout of step spillway was shown in Figure 1.

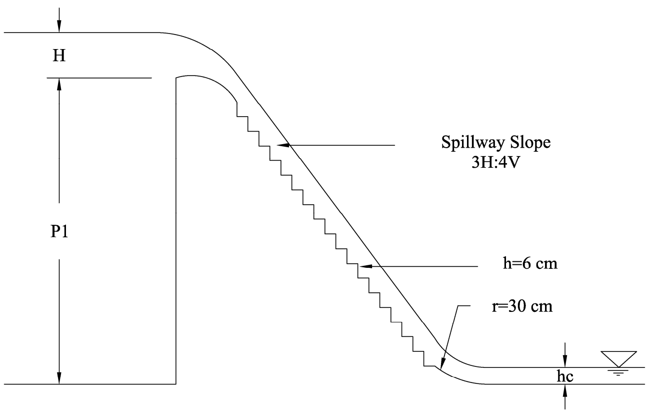

Figure 1. General layout of the model ,
No. of step=18 and step high=6 cm (Yani min et al 2001)

Theoretical calculation of hydraulic parameters

Three hydraulic parameters/characteristics of stepped spillway such as , damping ration “K”, resistance coefficient “ζ” and velocity coefficient “φ” can be calculated using following equations:

$$K = E_0 - \frac{E_c}{E_0} \tag{1}$$

$$\zeta = \frac{2gE_0K}{V_c^2} \tag{2}$$

$$\varphi = \frac{1}{(1+\zeta)^{0.5}} \tag{3}$$

where $E_0 = H_0 + \frac{V_0^{\ 2}}{2g}$ is energy head at upstream of the spillway, V_0 is approach velocity;H_0 is water depth at upstream of spillway; E_c = $h_c + \frac{V_c^2}{2g}$ is energy head at shrinkage section; V_c is velocity at shrinkage section and h_c is water depth at shrinkage section. In the step spillway energy loss (H_J) can be expressed as:

$$H_J = \zeta\left(\frac{V_c^2}{2g}\right) \tag{4}$$

Numerical modeling

The FLUENT (2006) was used as a CFD (computational fluid dynamics) solver in the present study to solve the three-dimensional Reynolds-averaged Navier-Stokes equations for incompressible flow. FLUENT solves the governing equations sequentially using the control volume method. Two-phase domain (water and a region of air on the top) is solved using FLUENT’s multiphase formulation, called the volume of fluid (VOF) method in order to resolve the variation of the water surface over spillway. The base of Navier-Stokes equations is on the preservation of momentum and mass within a moving fluid. The momentum differential equation is described by:

$$\frac{\partial}{\partial x}\left(\rho\vec{v}\right) + \nabla.\left(\rho\vec{v}\vec{v}\right) = -\nabla\rho - \nabla\left[\mu\left(\nabla\vec{v} + \nabla\vec{v}^{\,T}\right)\right] + \rho\,\vec{g} + \vec{F} \tag{5}$$

where P is pressure; $\vec{v}$ is velocity and g is gravity. The $\mu = \mu_0 + \mu_\tau . \mu_0$ is viscosity of fluid; μ_τ is the turbulence viscosity and $\vec{F}$ is the body force. The differential equation for mass conservation is described as below:

$$\frac{\partial}{\partial x}(\rho) + \nabla.\left(\rho\vec{v}\right) = 0 \tag{6}$$

where $\vec{v}$ is velocity and ρ is the fluid density. The K- ε model is an effective numerical simulation method which used in recent decades for simulation of turbulence. This method have been verified by experimental and field data. In this work K- ε was used to simulate the turbulence over the step-spillway.

Domain Description

As mentioned earlier, a two-phase domain containing flow on the spillway with a region of air at the top of flow solved using the multiphase method. The depth of air portion should be large enough to avoid any effect from the boundary condition at the top of the domain. If the ratio of air depth to the depth of water (critical depth) be one-third or larger, there is no effect from the boundary at the top of the domain (Salaheldin et al. 2005). In the present study, this ratio was set to be larger than 0.5. Although the convergence and stability of the solution were found to be insensitive to the grid size away the solid boundaries, but finer mesh is required near solid boundaries including the walls and steps in order to resolve the flow details near the solid boundaries (Figure 2). Accurate representation of flow near the wall region leads to a successful prediction of the turbulent flow away the walls and an accurate calculation of the bottom shear stress. The size of the cells adjacent to solid boundaries was chosen to satisfy the limitation of the wall unit distance $11.225<Y^+ <30$.

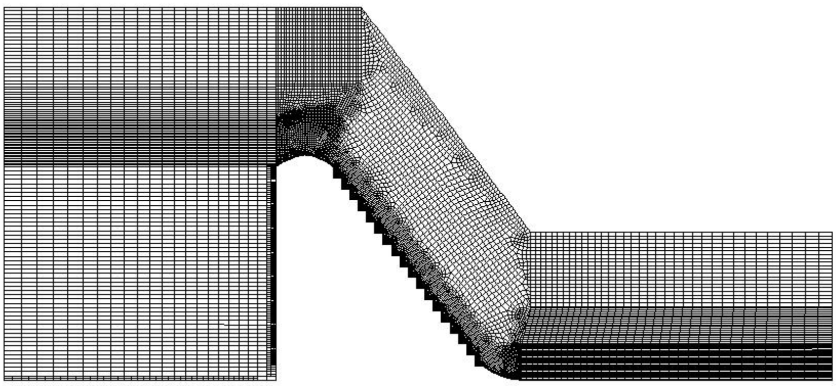

Figure 2. Mesh domain for step- spillway

The wall unit distance Y^+ is defined by the following equation:

$$Y^+ = \frac{\rho u_* y_p}{\mu} \tag{7}$$

where ρ is fluid density; u_* is shear velocity; y_p is distance from Point P to the wall and μ is dynamic viscosity of the fluid. Finer grid was also provided near the air-water interface to capture the small variation of the free surface. To model the spillway, a unstructured and structured meshes were used in different part of spillway. The unstructured mesh was used on spillway's steps because of complexity in geometry and in other part of domain, a structural mesh was applied. Based on trial and error, number of cell to generate the domain was determined to be equal to 22818 (Figure 2).

Boundary Conditions and Near Wall Treatment

Based on the nature of flow, proper conditions must be specified at the boundaries. In the present modelling, two separate boundaries were specified as inlets for air and water. At each inlet, the uniform distributions were assigned for all variables. In outlet, the outflows condition was given for both air and water. No-slip boundary was specified at solid boundaries to set the velocity to be zero.At the downstream boundary, an overall mass balance correction and zero normal gradients (zero diffusion flux) were defined for all flow variables. The water level is not defined in outlet and allowed to change as the hydrodynamic pressure. Symmetric boundary condition was assigned for the top surface which can be applied zero normal velocity and zero normal gradients for all variables (Fig. 3).

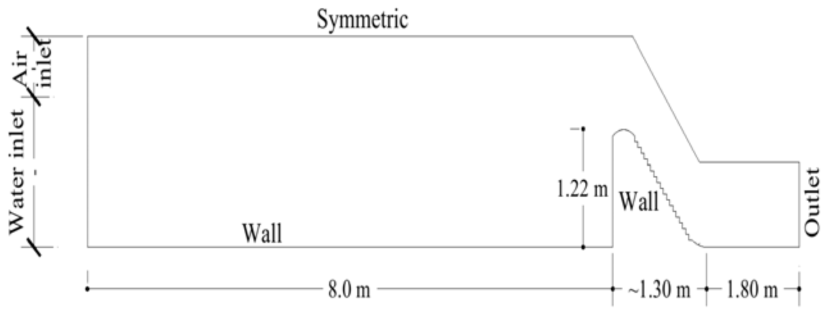

Figure 3: Boundary condition and dimension of model

Results and discussion

Fig. 4 shows two phase of water and air which obtained from numerical modelling. The VOF method separates both air and water with high accuracy. The stepped spillway dissipates a high rate of energy by interaction between the skimming flow and the vortices or eddies. Generally, the velocity on the spillway was used to design the energy dissipater in downstream of spillway. The size of the energy dissipater can be designed based on the residual energy on the stepped spillway, the residual energy or energy ratio can be calculated according to the kinetic and potential energy upstream and downstream of the spillway.

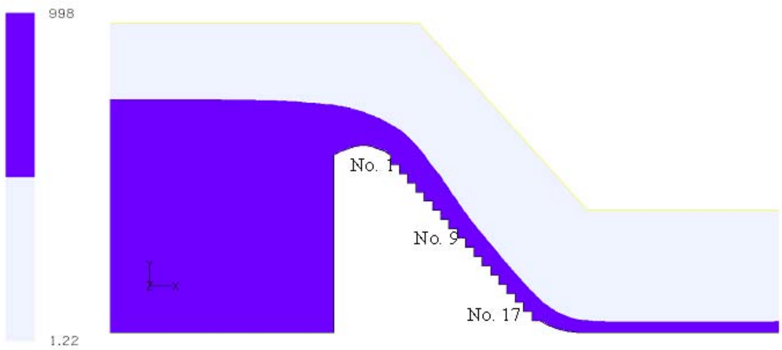

Figure 4: Free water surfaces obtained by simulation

In Fig.5,the velocity vector plots at three different location of the crest, immediately after the crest (case-a), at middle (case-b) and before the toe (case-c) .As shown in this figure , the skimming flow is developed over the step edges and at triangular corner of each step, the vortices (recirculation flow) formed in clockwise direction. As can be seen, the centre of the eddy is near to vertical wall and above the horizontal surface of the steps. The velocity legend in Fig. 4 shows, the magnitude of the recirculation flow velocity is reduced by about 1/3 than that of the skimming flow.

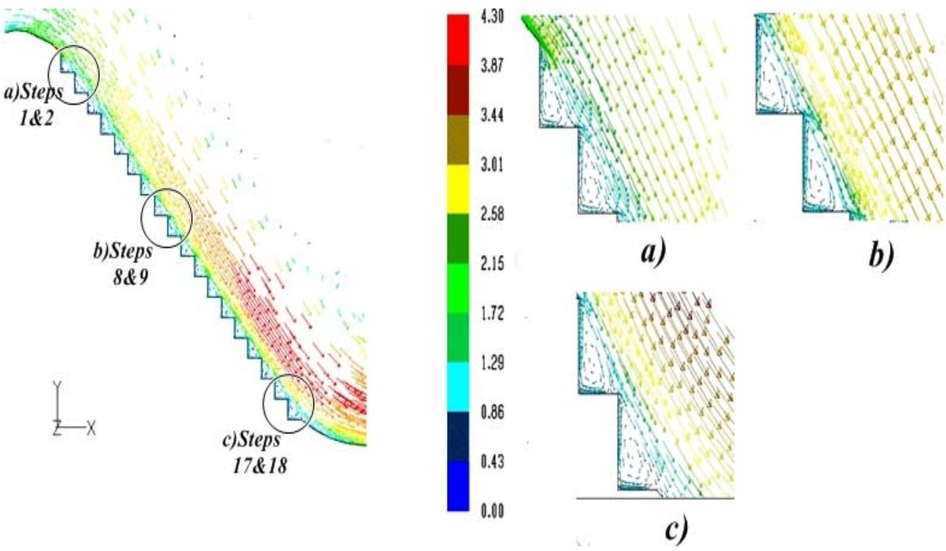

Figure 5: Velocity vectors (m/s) on (a) After the crest, steps 1&2;

(b) At middle of spillway, steps 8&9; (c) Before the toe steps 17&18

In the spillway, the potential of cavitation and vibration increase if negative pressures develop at certain locations of spillway. In order to study the pressure distribution within a step and its variation from first step to last one, the pressure profiles at three locations, after crest, spillway middle and before the toe were investigated in Fig. 6 and 7. In this figures both X and Y coordinate have been normalize using step length (L_s) and step high (h_s) respectively. Irrespective of the step location, similar pattern occur for pressure distribution over the x and y coordinates of the step surfaces.

As shown in Fig. 6, with increasing distance away from the corner (x-coordinate or horizontal surface), the pressure decreases to reach a minimum level and then increases to arrive at maximum level, afterward falls down dramatically at the tip of step. Fig. 6 shows the minimum pressure occurs before the middle of each step, between 0.2 to 0.4 and the maximum pressure was located after the middle at $X/ L_s \geq 0.85$. This maximum pressure is caused by the impact of the falling water on the steps. Therefore in the stepped-spillway, the pressure on the last step (step-17) which located near to toe is greater than that first step (step-1).

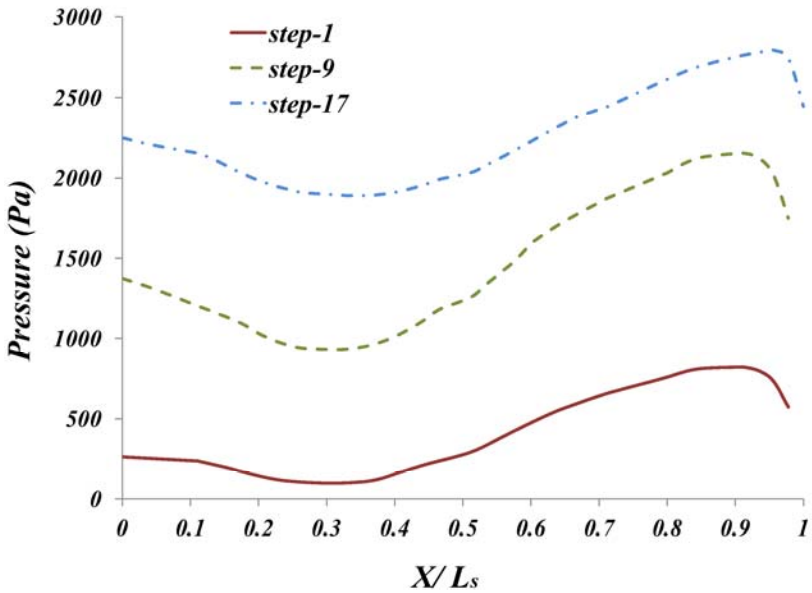

Figure 6: Pressure profiles on horizontal surface of steps

Along the vertical surface (Fig.7), the pressure decreases gradually with increasing distance upward from the corner (y-coordinate), afterward a sharp variation occur close to step tip. Fig. 7 indicates that the minimum pressure occurs near to step tip and in range of $0.7 \leq Y/h_s \leq 1.0$.

The results in vertical edge of steps show the minimum pressure is depends on step location, no negative pressure can be seen on last step (step-17) whereas, the maximum mines pressure occur on first step (step-1). Therefore the probability of cavitations decreases from crest to toe of spillway.

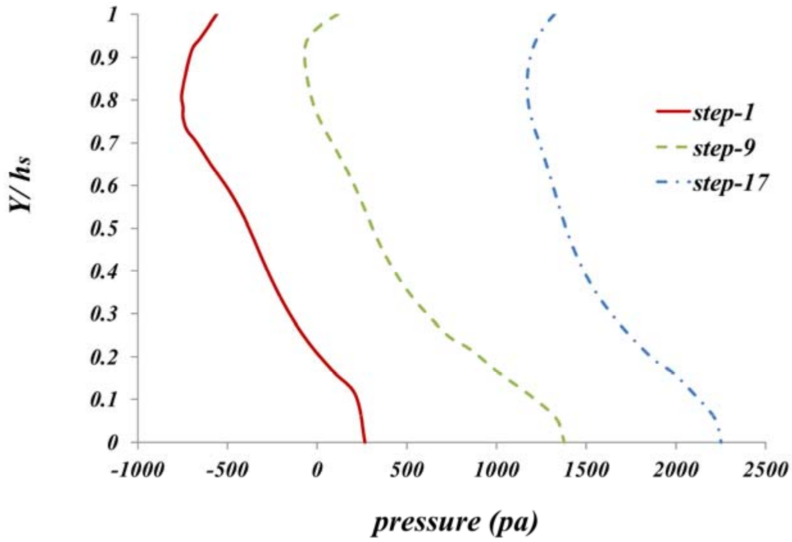

Figure 7: Pressure profiles on vertical surface

As mentioned in equation(4), the energy loss over a spillway is a function of damping ration "K" and resistance coefficient "ζ". In this study, all three hydraulic parameters of K, ζ and φ were obtained using the numerical modelling. The comparison between numerical and experimental data for hydraulic characteristics was shown in Fig. 8. To validate the numerical results, three common error measures is used (Table 1) namely, coefficient of determination (R^2), root mean square error (RMSE) and mean absolute error (MAE). As shown in this table a good agreement occurred for three of φ , k and ζ , with high value for R^2 (0.92 , 0.96 and 0.96 respectively) and low value for RMSE (0.034 , 0.041 and 0.096 respectively). Statistical analysis shows, numerical modelling can predict three hydraulic characteristics with high accurate value.

Table 1: Analysis of sensitivity for independent parameters

FUNCTION	R^2	*RMSE*	*MAE*
velocity coefficient "φ"	*0.92*	*0.034*	*0.029*
damping ration "K"	*0.96*	*0.041*	*0.036*
resistance coefficient "ζ"	*0.96*	*0.096*	*0.082*

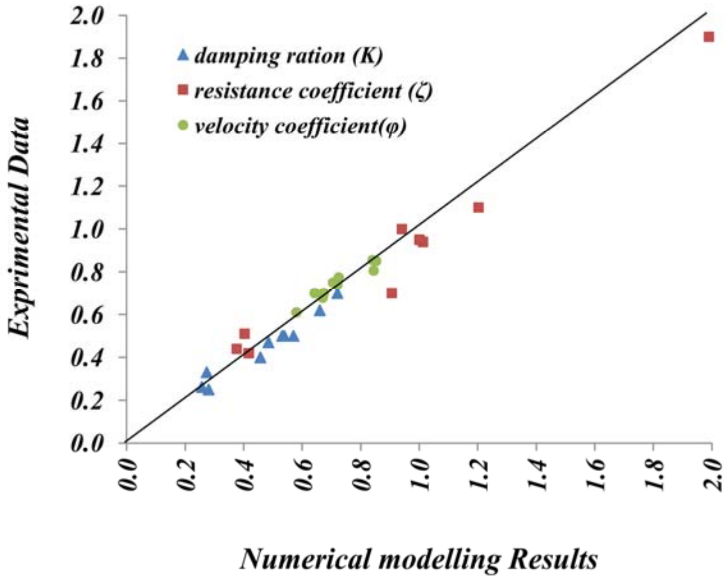

Figure 8: Comparison between numerical and experimental data

Conclusion

In this research, the numerical modelling was used to predict the pressure and energy loss characters over the WES spillway with similar steps. The energy loss over the step spillway is a function of hydraulic parameter of damping ration "K" and resistance coefficient "ζ". The methods of VOF and K- ε were applied to simulate the free-surface water and turbulence respectively. Both structured and unstructured meshes were used to generate the domain; the difficulty in treating the complex boundaries of stepped spillway was overcome by using an unstructured grid. The result showed in horizontal surface of each step, the minimum pressure occurs before the middle of step ($0.2 \leq X/L_s \leq 0.4$) , and the location of maximum pressured have been after middle ($X/L_s \geq 0.85$).The positive pressure occurred in all horizontal surface, therefore the cavitations probability on horizontal surface is impossible. Along the vertical surface, the minimum pressure occurs near to step tip and in range of $0.7 \leq Y/h_s \leq 1.0$. The negative pressure on first step (near to crest) was greater than that for last step(close to toe) , therefore the probability of cavitation for steps near to crest is higher than that of steps close to spillway tip. The results of numerical modelling for three hydraulic characteristics ("φ", "K" and "ζ") were evaluated by statistical analysis. The comparison between prediction and observed data indicates that, the numerical simulation can predicate energy loss parameter with high accuracy. Due to less time demand and lower cost of the numerical method than that of experiments therefore numerical method has significant advantage in practical projects.

References

Bhajantri, M.R., Eldho, T.I., Deolalikar P. B. (2007). Modeling hydrodynamic flow over spillway using weakly compressible flow equations. Journal of Hydraulic Research, 45(6), 844-852.

Boes, R.M., Hager, W.H. (2003). Two-Phase Flow Characteristics of Stepped Spillways. J. Hydraulic Eng. 129(9), 661-670.

FLUENT user's guide manual-version 6.3 (2006) Fluent incorporated,Lebanon. N. H.

Salaheldin, T. M., Imran, J.,Chaudhry, M. H. (2004) "Numerical Modeling of Three-Dimensional Flow Field Around Circular Piers" Journal of Hydraulics Engineering, ASCE. pp. 91-100.

Johnson, M. C. ,Savage, B. M. (2006) Physical and numerical comparison of flow over ogee spillway in the presence of tailwater. Journal of Hydraulic Engineering-Asce, 132, 1353-1357.

Olsen, N.R.B., Kjellesvig H.M. (1998). Three-dimensional numerical flow modelling for estimation of spillway capacity. Journal of Hydraulic Research 36(5), 775 - 784.

Yang, M., Gao, J. H., Lian, J. J. (2001) Study on hydraulic characteristics of stepped spillway. Hydraulics of Rivers, Water Works and Machinery, Vol 1, Theme D, Proceedings, 856-861.