Malaysian Journal of Civil Engineering 30(1):57-68 (2018)

NUMERICAL STUDY OF TURBULENT FLOW FOR MODERATE-SLOPE STEPPED SPILLWAYS

Bentalha Chakib*

Department of Hydraulic, Faculty of Technology, Abou Bakr Belkaid University- Tlemcen, Algeria

*Corresponding Author: *c_bentalha@yahoo.fr*

Abstract: Stepped spillway is a power full hydraulic structure for energy dissipation because of the large value of the surface roughness. The performance of the stepped spillway is enhanced with the presence of air that can prevent or reduce the cavitation damage. This work aims to simulate air entrainment and determine the characteristics of flow at stepped spillways. Within this work flow over stepped chute is simulated by using fluent computational fluid dynamics (CFD). The volume of fluid (VOF) model is used as a tool to simulate air-water interaction on the free surface thereby the turbulence closure is derived in the $k-\varepsilon$ turbulence standard model. The found numerical results agree well with experimental results.

Keywords: *Air-water flow, fluent, VOF model, stepped spillway, standard $k-\varepsilon$ model*

Article history:
Received 31 January 2017
Received in revised form 12 June 2017
Accepted 24 October 2017
Published online 30 April 2018

1.0 Introduction

A spillway is a structure designed to discharge the surplus flood water from the dam. Evacuating the surplus of water to stalling basin creates high levels of kinetic energy. This energy may be achieved by construction of steps on the spillway (Chanson, 2001a).The energy dissipated with steps was two to three times as great as the energy dissipated with a smooth surface (Charles and Kadavy, 1996) and can reduce the size of stilling basin at the toe of the dam (Rajaratnam,1990, Christodoulou, 1993 and Chanson, 2001b). The performance of the stepped spillway is the presence of air which can prevent or reduce the cavitation erosion damage. The skimming flow regime down stepped chute is characterised by highly turbulence and the water flows as a coherent stream.

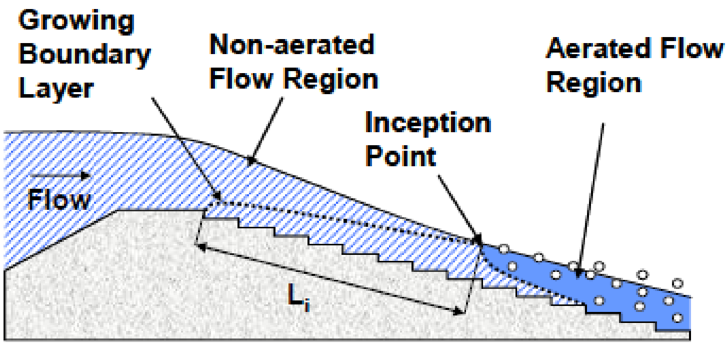

Figure 1: Position of the inception in stepped spillway

In the skimming flow regime, air entrainment occurs when the turbulent boundary layer thickness coincides with the water depth (Chanson, 2001b). This location is called the inception point (e.g. Figure 1). At the inception point upstream, the flow is smooth and glassy whereas at the downstream of the inception point the flow becomes uniform as the depth of the air-water mixture grows.

The topic of the flow over stepped spillway was the object of several experimental works, today, with the use of high-performance computers and more efficient computational fluid dynamics (CFD) codes, the flow over spillways can be investigated numerically in reasonable time and with reasonable expense (Chinnarasri *et al.*, 2012). Chen *et al.* (2002), simulated flow over a stepped spillway using the k –ε turbulence model. Bombardelli *et al.* (2011) Simulated non-aerated region of the skimming flow in steep stepped spillways using 3D-FLOW. Numerical study of Mohammed *et al.* (2012) was performed to simulate and investigate flow characteristics over a steeply sloping stepped spillway. They used VOF model to simulate interaction between air and water and Turbulence was encountered by both RNG k-ε and Large Eddy Simulation (LES). Afshin and Mitra (2012) used FLUENT commercial software for examining the performance of the volume of fluid (VOF) and mixture models in simulating skimming flow over stepped spillway Cheng *et al.* (2006) used a VOF model to investigate the normal velocity profiles responding to the steps as bed roughness. The RNG (k- ε) model was chosen and their numerical results successfully reproduced the flow over the stepped spillway of the physical model.

This study, present the results of numerical simulation flow in moderate slope stepped spillway obtained by using Fluent computational fluid dynamics (2006) to show the distribution of velocity, pressure and turbulent kinetic energy along the stepped spillway. Likewise the cavitation damage has been discussed in this work. The simulation results were compared with the experimental data of Felder and Chanson (2009).

2.0 Numerical Model

Fluent computational fluid dynamics (CFD) is used to solve Navier-Stokes equations that are based on momentum and mass conservation of multi-phase flow over stepped spillway. Because the standard $k - \varepsilon$ model is still a good tool for numerical simulation of flow in stepped spillways and verified by experimental and field data (Chen *et al.*, 2002), it is used to simulate turbulence.

Continuity equation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_i}{\partial x_i} = 0 \tag{1}$$

Momentum equation:

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \rho g_i + \frac{\partial}{\partial x_j}\left\{(\mu + \mu_t)\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right\} \tag{2}$$

Turbulence kinetic energy equation (k):

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right] + G_k - \rho\varepsilon \tag{3}$$

Turbulence dissipation rate energy equation (ε):

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right] + C_{\varepsilon 1}\frac{\varepsilon}{k}G_k - C_{\varepsilon 2}\rho\frac{\varepsilon^2}{k} \tag{4}$$

Where, G_k is production of turbulent kinetic energy which can be given as

$$G_k = \mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j} \tag{5}$$

μ_t is the turbulent viscosity that satisfies

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{6}$$

C_μ=0.09 is a constant determined experimentally;
σ_k and σ_ε are turbulence Prandtl numbers for k and ε equation respectively, σ_k =1.0, σ_ε=1.3,
$C_{1\varepsilon}$ and $C_{2\varepsilon}$ are ε equation constants, $C_{1\varepsilon}$=1.44, $C_{2\varepsilon}$=1.92.

The volume of fluid (VOF) method is applied to simulate the free surface between water and air (Fluent 2006). In this approach, the tracking interface between air and water is accomplished by the solution of a continuity equation for the volume fraction of water:

$$\frac{\partial \alpha_w}{\partial t} + \frac{\partial \alpha_w u_i}{\partial x_i} = 0\,;0 \le \alpha_w \le 1 \tag{7}$$

Where, α_w is volume fraction of water.

In each cell, the sum of the volume fractions of air and water is unity. So, volume fractions of air denote α_a can be given as

$$\alpha_a = 1 - \alpha_w \tag{8}$$

The geometry of numerical model and boundary conditions are shown in figure 2.The channel was 3.2 m long and 1 m wide. The channel slope is 21,8°. The stepped spillway contains 20 identical steps with 0.05m height and 0.125 m length by step. The two-dimensional numerical domain was divided into unstructured grids (triangular cell) that had a high adaptability to the complex geometry and boundary. Triangular meshes with 0.015 m^2 are used.

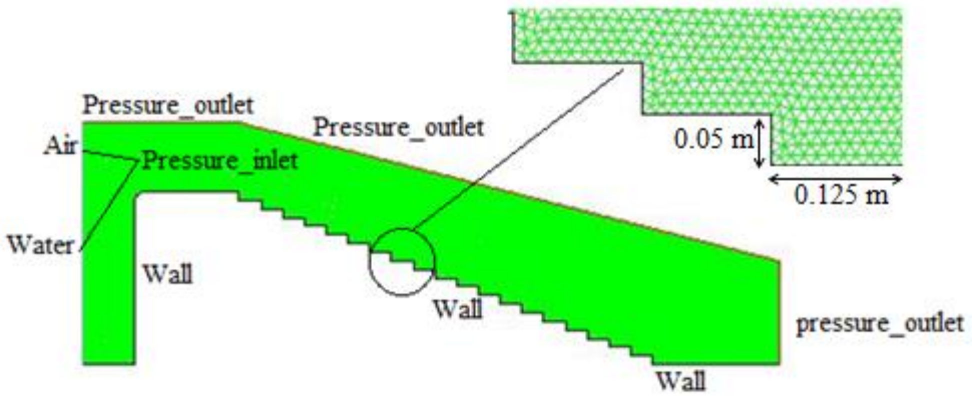

Figure 2: Boundary conditions and numerical model of a stepped spillway

The boundary conditions in this study are pressure inlet as water inlet and air inlet, outlet as a pressure outlet type. All of the walls as a stationary, no-slip wall. The viscosity layer near to the wall dealt with the standard wall function. The boundary conditions for the turbulent quantities such as k and ε can be calculated from (Fluent 2006):

$$k = \frac{3}{2}\left(U_{avg} I\right)^2 \tag{9}$$

$$\varepsilon = C_u^{3/4} \frac{k^{3/2}}{0.07 D_H} \tag{10}$$

Where, *I* is turbulence intensity can be estimated from the following formula derived from an empirical correlation for pipe flows:

$$I = 0.16\left(Re_{DH}\right)^{-1/8} \tag{11}$$

U_{avg} is the mean velocity of water flow inlet and D_H is the hydraulic diameter.

3.0 Results and Discussion

In this study, the position of the inception point are computed and compared with the existing experimental results (Felder and Chanson 2009).

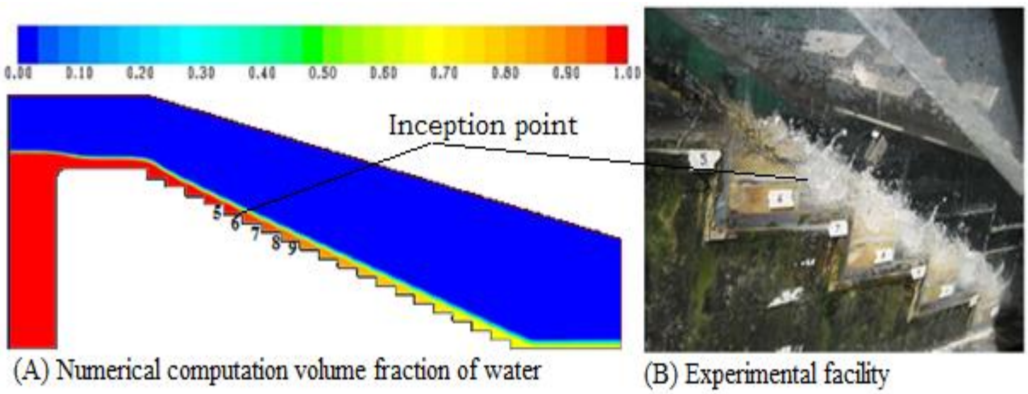

Figure 3: Measured and computed inception point for q=0.0436m^2/s

Figure 3 presents a comparison the air entrainment simulated by VOF model and in experiments for unit discharge equal to 0.0436m^3/s . Good agreement between observed and computed inception point from this figure. Figure 3 indicates that, the air entrainment started on step edge N°6. At the inception point, the degree of turbulence was large enough to entrain air into the black water flow (Cheng 2006), and then the volume fraction of water becomes less than unity.

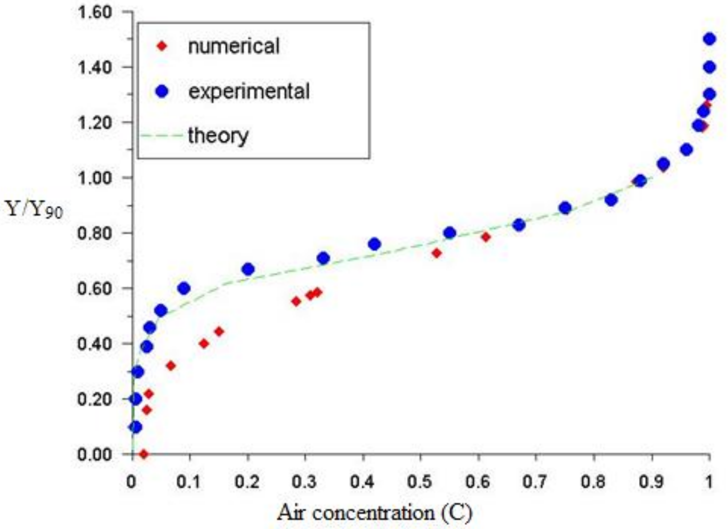

Figure 4: Experimental and computational air concentration distribution compared with equation (12)

Downstream of the inception point of free-surface aeration, air and water are fully mixed, forming a homogeneous two-phase flow. In skimming flow, the air concentration distribution may be described by an analytical solution of the air bubble advective diffusion equation:

$$C = 1 - tanh^2\left(K'' - \frac{\frac{y}{Y_{90}}}{2D_0} + \frac{\left(\frac{y}{Y_{90}} - \frac{1}{3}\right)^3}{3D_0}\right) \qquad (12)$$

where y is distance measured normal to the pseudo-invert, Y_{90} is the characteristic distance where C = 90%, K is an integration constant and D_o is a function of the of the mean air concentration C_{mean} (Chanson,2001b). In figure 4, the computed air concentration profile downstream of the inception point is compared with Felder and Chanson experiments (2009) and with equation (12) for q= 0.122 m^2/s. The relative error between the numerical and experimental results is 1.2% to 14.8% except at y/Y_{90} ranging from 0.3 to 0.6; the relative error is slightly higher. This increase in error may be due to the VOF model which underestimates the value of air concentration (Afshin and Mitra, 2012). Equation 12 compares favourably with the numerical and experimental air concentration profiles. It is obvious that numerical model is capable of computing air concentration in such regions (Mohammed *et al.*, 2012).

The comparison of velocity between simulation and measurements in Felder and Chanson (2009) are shown in figure 5 for q=0.122 m^2/s. The percentage difference between numerical and experimental data was less than 10%, which shows good agreement, except at y/Y_{90} ranging from 0.1 to 0.2; the relative error is slightly higher. This increase in error may be stem from the three-dimensional structure of the flow and the anisotropy is not captured in 2D (Bombardelli *et al.*, 2011).

In the skimming flow, the velocity profile increase from the pseudo bottom at the free surface flow. Based on previous studies (Chanson, 2001b and Boes and Hager, 2003), the distribution of air-water velocity follows a power law given by:

$$\frac{V}{V_{90}} = \left(\frac{y}{Y_{90}}\right)^{1/n} \tag{13}$$

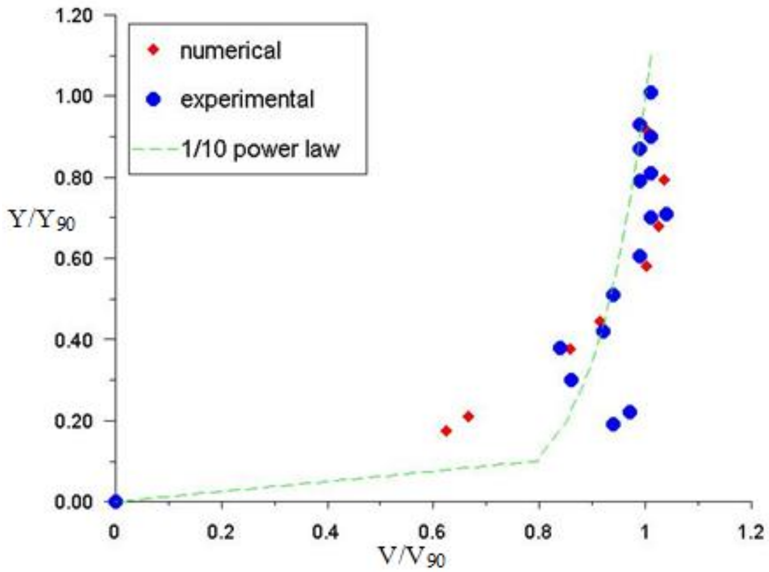

Figure 5: Dimensionless velocity at step edge 18 for q=0,122 m^2/s

Where V_{90} is the characteristic velocity for C = 90%.The exponent n is obtained from experiments data. From figure 5, the velocity profiles tended to follow a 1/10 power law distribution.

The contour of velocity in stepped spillway is shown in figure 6. The velocity is higher in aerated region because the entrained air reduces wall friction; also the fluid is accelerated by the gravity along the chute. This result is qualitatively similar to those presented by Quian *et al.* (2009).

The recirculation flow which dissipates the energy in step corner is presented in figure 7. Most of the energy is dissipated by momentum transfer between the skimming flow and the eddy in the interior of the step.

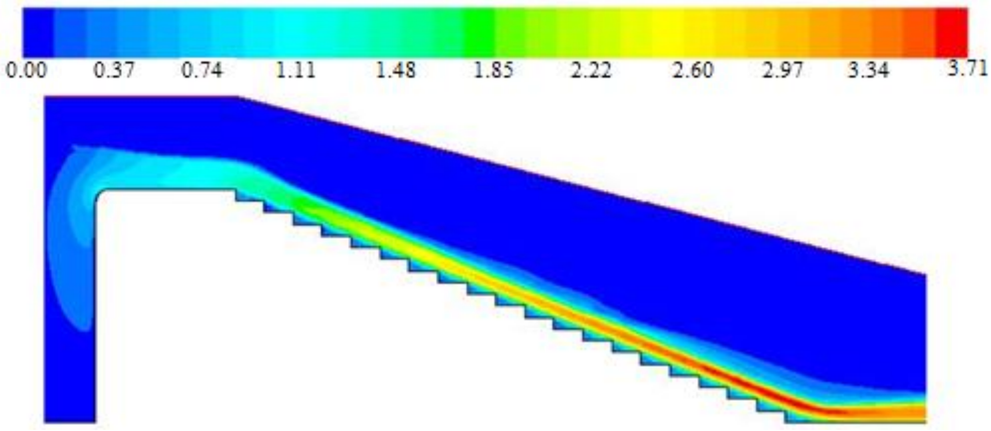

Figure 6: Velocity distribution along the stepped spillway for $q=0.122 m^2/s$

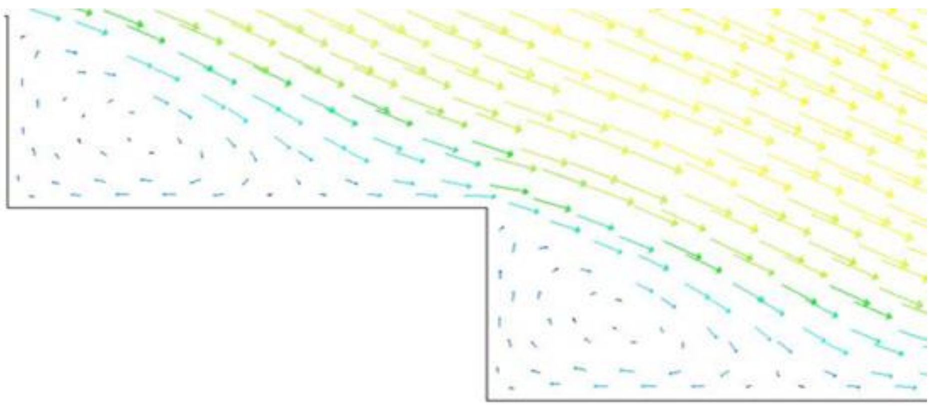

Figure 7: Recirculating vortices in the triangular zone of the step corner for $q=0.122 m^2/s$

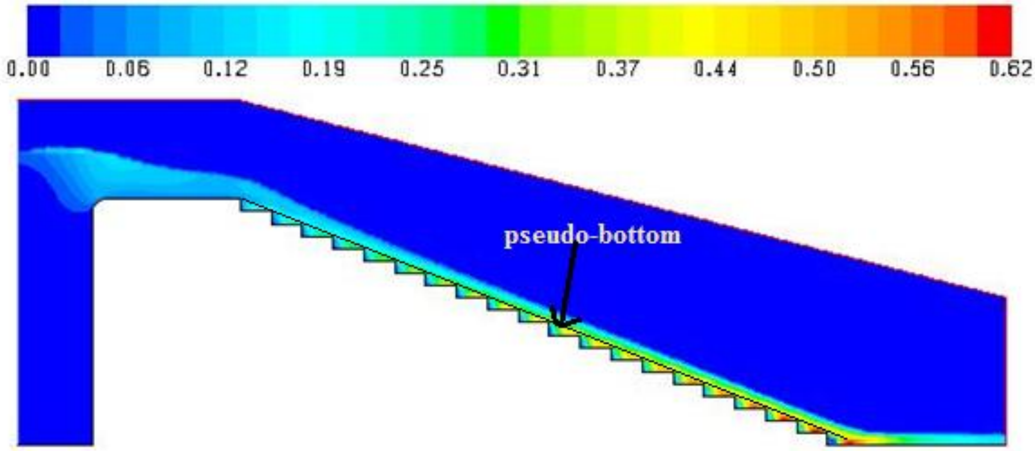

Figure 8: Turbulence kinetic energy along the stepped spillway for q=0.122m^2/s

Figure 8 present the distribution of kinetic turbulent energy (k) along stepped spillway for q=0.122m^2/s. As can be seen from this figure, the maximum of k is located near the pseudo-bottom as a region of formation and growth of recirculating vortices. Also it can be observed the increasing of turbulent kinetic energy along the stepped spillway which is the result of the development of the boundary layer. A gradual decrease of *k* is observed near the water surface, where the flow can be considered irrotational. This result is qualitatively similar to those presented by Bombardelli *et al.* (2011).

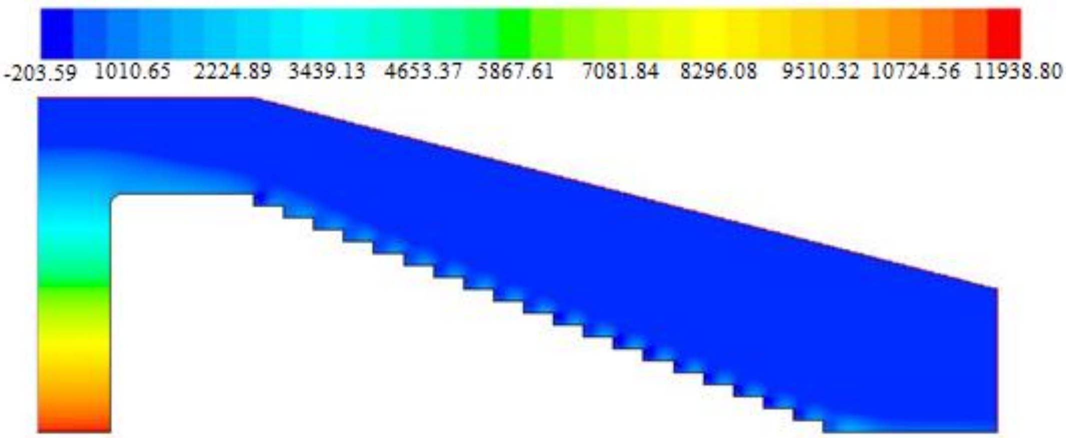

Figure 9: Contour of static pressure along the stepped spillway for q=0.122m^2/s

Figure 9 display the contour of static pressure for q=0.122 m^2/s. This figure indicate that the minimum value of pressure is located in the vertical wall of the step, is due by separation flow between skimming flow and the eddy in this region. Also maximum pressure exists in the horizontal surface of the step. This maximum pressure is caused by the impact of the skimming flow coming from upper step.

The risk of cavitation appears for high velocity flows and when the values of static pressure becomes lower then vapour pressure of water, resulting in the local cavitation index:

$$\sigma = \frac{P - P_V}{\rho \frac{V^2}{2}} \tag{14}$$

Where V is reference flow velocity, P is reference flow pressure, P_V is water vapour pressure and ρ is water density. To prevent the cavitation damage to a stepped spillway, it is required to keep σ more than cavitation inception index σ_i. Based on the study of Falvey (1990), the cavitation inception index varies between 0.076 and 1.2 for all surfaces roughness. Matos et *al.* (2014) proposed formulae to evaluate the damage of cavitation at the inception point:

$$\sigma = 0.064 F_*^{-0.23}\left[1 + \frac{4.762\left(\frac{P_{atm} - P_V}{\rho g}\right)}{k_s F_*^{0.59}}\right] \tag{15}$$

Where:
$k_s = h\ cos\theta$
h = step height
θ = channel slope
F_* = Froude number defined in terms of the roughness height:
$F_* = q/[g(sin\,\theta)\{k_s\}^3]^{0.5}$
q = unit discharge
g = gravitational constant

In figure 10, equation 15 is plotted as function F_* for step height equal 0.05 m and 0.1m. This figure shows that, the risk of cavitation increase with increasing of F_* therefore when the flow rate increase, also with growing of step height.

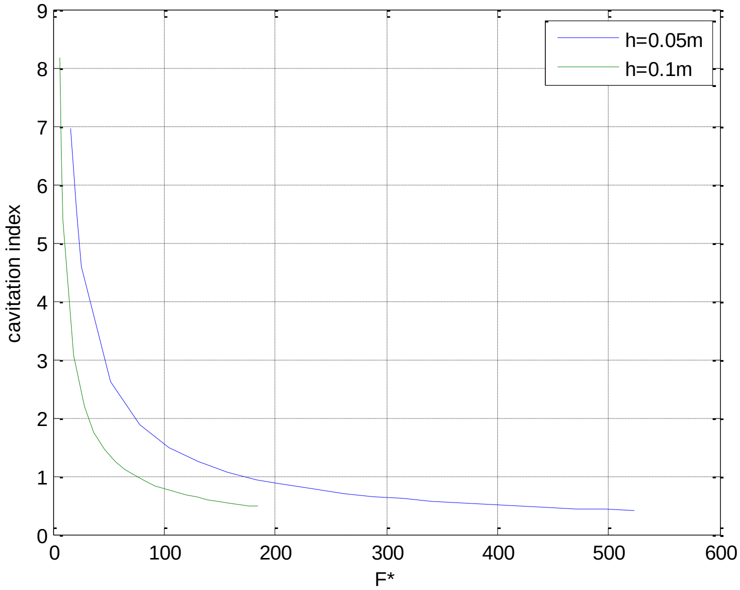

Figure 10: Cavitation index versus Froude surface roughness, F_*

4.0 Conclusion

In the present numerical study, flow over stepped spillway was simulated by using fluent. Free surface was treated by VOF model and turbulence flow was estimated by k-ε Standard Model. The experimental studies of Felder and Chanson (2009) are used to validate the numerical results of the air-water two-phase flow over the stepped spillway. Good agreement is found between numerical and experimental results except at y/Y_{90} ranging from 0.3 to 0.6; the relative error is slightly higher between numerical and experimental air concentration profile. Also at y/Y_{90} ranging from 0.1 to 0.2; the relative error is somewhat higher between computed and measured velocity profile. It was found that the calculated inception point is well agreed with that of measurement. It has been verified that the velocity profile follows the one-tenth power law distribution and the maximum of kinetic energy of turbulence is found near the pseudo-bottom. Minimum and maximum pressure is located in the vertical and horizontal face of step. Finally the risk of cavitation damage increase by increasing of flow rates and step height.

References

Afshin Eghbalzadeh and Mitra Javan, “Comparison of Mixture and VOF Models for Numerical Simulation of Air–entrainment in Skimming Flow over Stepped Spillways” ,J. of Science Direct. Procedia engineering , (2012)

Boes, R. M. and W. H. Hager, “Two-phase flow characteristics of stepped spillways”. Journal of Hydraulic Engineering, ASCE,129, N°9 (2003), pp 661-670.

Bombardelli FA, 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. Environ. Fluid Mech,, 2011, 11(3),263-288.

Chanson. H.” Air Bubble Entrainment in Free-Surface Turbulent Shear Flows”. Academic Press, London, UK, (1997)

Chanson, H “The Hydraulics of Stepped Chutes and Spillways.” Balkema Publ., Rotterdam, The Netherlands. (2001a)

Chanson, H. "Hydraulic Design of Stepped Spillways and Downstream Energy Dissipators." Dam Engineering,.11, N°4(2001b) , 205-242

Charles E.Rice and Kem C.Kadavy., Model of A roller compacted concrete stepped spillway. Journal of Hydraulic Engineering, ASCE, 122N°6 (1996), 292-297.

Cheng Xiangju, CHEN Yongcan and LUO Lin,, Numerical simulation of air-water two-phase flow over stepped spillways. Science in China Series E, Technological Sciences, ,49 N°6 (2006) ,674-684

Chen, Q., G.Q. Dai and H.W. Liu., Volume of Fluid Model for Turbulence Numerical Simulation of Stepped Spillway Over Flow, Journal of Hydraulic Engineering, ASCE, 128 N°7(2002), 683-688.

Chinnarasri. C, Kositgittiwong.K and. Julien. PY” Model of flow over spillways by computational fluid dynamics”, Proceedings of the Institution of Civil Engineers,(2012)

Christodoulou,G.C., Energy dissipation on stepped spillways. Journal of Hydraulic Engineering, ASCE, 119 N°5, (1993), 644-650.

Falvey, H. T.” Cavitation in chutes and Spillways”. Engineering monograph No. 42, United States Department of the Interior – Bureau of Reclamation, Denver, Colorado, (1990).

Fluent, Manuel and user guide, Fluent Inc, 2006.

Matos J. and Meireles I. “ Hydraulics of stepped weirs and dam spillways: engineering challenges, labyrinths of research” 5th International Symposium on Hydraulic Structures, Brisbane, Australia, 25-27 June 2014.

Mohammad S, Jalal. A and Michael. P “Numerical Computation of Inception Point Location for Steeply Sloping Stepped Spillways” 9th International Congress on Civil Engineering, Isfahan University of Technology (IUT), (2012).

Qian ZhongDong, HU XiaoQing, Huai WenXin and Amador António.” Numerical simulation and analysis of water flow over stepped spillways”. Science in China Series E: Technological Sciences, 52 N°7(2009),1958-1965

Rajaratnam, N. “Skimming flow in stepped spillways”. Journal of Hydraulic Engineering, ASCE, 116 N°4 (1990), 587-591.

Stefan Felder and Hubert Chanson. “Energy dissipation, flow resistance and gas-liquid interfacial area in skimming flows on moderate-slope stepped spillways” Environ Fluid Mech (2009) 427–441