Two-phase Flow over Stepped and Smooth Spillways: Numerical and Physical Models

Duangrudee Kositgittiwong, Chaiyuth Chinnarasri, and Pierre Y. Julien

Abstract –**Numerical modeling of stepped spillways is very complicated and challenging because of the high roughness and velocity recirculation regions. Two types of multiphase flow models are used: a mixture multiphase flow model (MMF) and a volume of fluid multiphase flow model (VOF). In both models, the realizable k-ε model is chosen to simulate turbulence. The computational results are compared with large-scale experimental data from Colorado State University. The spillway was 1.22m wide and consisted of smooth, 25, and 50 horizontal steps. The discharge was varied from 0.20 to 3.28 m^3/s. The data series obtained for model comparison include; velocity profiles, energy dissipation, and characteristics of flow. Both models can satisfactorily simulate the flow pattern and the recirculation regions. The velocity profiles are more accurately simulated using the VOF model.**

Keywords – **Smooth spillway, Stepped spillway, Numerical model, Physical model.**

I. Introduction

In recent years, there has been an increase in the rate of surplus or flood waters which cause high flow into the reservoirs. Due to the increase of water flow, the storage capacity may be exceeded. The dam spillways must be designed to release surplus or flood water and to avoid exceeding reservoir capacity. Gonzalez and Chanson [1] mentioned higher design flows that affect the insufficiency of the existing spillway capacity. A spillway is kind of hydraulic structure that is provided at storage and detention dams to release water that cannot be safely stored in the dam [2]. To be safe, the spillway must be capable of passing high flow without jeopardizing the dam. A stepped spillway is an important kind of spillway having profile made up of steps. It dissipates much more energy than other types of spillways when water is flowing over the spillway profile. According to Chanson [3], stepped spillways have been used for at least 1500 years. Historically, very active experimental research has been done on the air-water flow over stepped spillways, such as flow patterns, inception of air entrainment, air concentration, velocity field, pressure field and energy dissipation [4]. The engineers have normally investigated the flow through laboratory experiments on scaled down models of spillways. The complexity of the flow structure which includes complicated boundary conditions, the curved free surface, and the unknown scale effects has caused uncertainties in transposing the experimental results to prototype scales. With the development of computational fluid dynamics (CFD) and high-performance computers, complex multiphase flows can be simulated numerically and with validation the results can be trusted to be reliable.

Manuscript received August 15, 2012. This work was supported by the Thailand Research Fund/Royal Golden Jubilee PhD Grant (TRF/RGJ) under grant number PHD/0225/2548 and BRG5280001 and the National Research University Project of Thailand's office of the Higher Education Commission.

D. Kositgittiwong is with Water Resources Engineering & Management Research Center (WAREE), Department of Civil Engineering, King Mongkut's University of Technology Thonburi, Bangkok 10140, Thailand. (e-mail: duangrudee.k@gmail.com)

C. Chinnarasri is with Water Resources Engineering & Management Research Center (WAREE), Department of Civil Engineering, King Mongkut's University of Technology Thonburi, Bangkok 10140, Thailand. (phone: +662-470-9136-7; fax: +662-427-9063; e-mail: chaiyuth.chi@kmutt.ac.th).

P. Julien is with Department of Civil and Environmental Engineering, Colorado State University, Colorado 80523, USA. (e-mail: pierre@engr.colostate.edu).

ISSN-1584-5990

Given reduced time demand and lower cost of the numerical method than physical experiments, simulation of the stepped spillway overflow has a significant advantage.

The two well established and widely used computational methods are the finite difference and finite element methods. Tabbara et al. [2] used the finite element method to predict stepped flows at the small scale of experiments. In the upper part of the flume as well as in the bottom part steps were introduced along the chute such that the envelope of their tips followed the smooth spillway chute profile. Although the results of this study are encouraging, physical or laboratory measurements are still crucial for providing reference data. Benmamar et al. [5] developed a numerical model for two-dimensional boundary layer flow over a stepped channel with steep slope, which was based on the implicit finite difference scheme. The finite volume method, which has been extensively used to model a wide range of fluid-flow problems, was originally developed as a special finite difference formulation. Qian et al. [4] used a MMF model to simulate flows over a stepped spillway. The turbulence models those were investigated are realizable k-ε model, SST k-ω model, v^2-f model and LES model. There were 40 steps with the step height of 0.05 m. The study region comprised the 6^{th} to 12^{th} steps from the crest. The realizable k-ε model show the best agreement in simulating flow over stepped spillways. Dong [6] studied numerical simulation of skimming flow over mild stepped channel. The channel consisted of 40 steps with the ramp angle, θ of 10^{o} and 20^{o}. All air boundaries were defined as pressure boundaries with zero pressure specified. Smooth channel flow was also simulated to compare the hydraulic characteristics of the stepped channel flow with the smooth one. Chen [7] used the k - ε turbulence model to simulate the complex turbulence overflow. Their first five steps were varied while the size of the rest were 0.06 m high and 0.045 m long. The study indicated that the turbulence numerical simulation is an efficient and useful method for the complex stepped spillway overflow.

Most of the previous studies focused on the small scale stepped spillways and tested discharge. Since 1966, the possible scale effects for steep spillways had been already mentioned [8]. Due to the viscosity and surface tension play an important role in highly turbulent air-water flow, so, only Froude similitude is not enough to study flow in stepped spillway without the scale effects. For the true similarity, the Froude similitude can be used when the step height >2 cm, Reynolds number $>10^5$ and Weber number ≥ 100 [9, 10, 11]. Therefore, large scale of stepped spillway is studied for both experiments and numerical model. Also due to less time demand and lower cost of the numerical method than that of experiments, this study will be emphasized on numerical model to attest that numerical model can be used compared with large scale experiments. The flow pattern and flow characteristics, flow profiles along the spillway, velocity profiles, and air concentration profiles, were collected from the experiments which consist of nappe flow and skimming flow. The VOF and MMF were used as a multiphase flow model to compare the better one using with large scale experiments.

2. Numerical Model

The stepped spillway was modeled as shown in Fig. 1. For each case studied, there are 230,565 quadrilateral cells with 232,109 nodes created. Quadrilateral meshes with 0.1×0.1 m^2 are used. The boundary conditions in this study are no-slip wall, outlet as a pressure outlet type, free surface as a pressure inlet type, air inlet and water inlet as a velocity inlet type. The inlet water velocity is set as uniform at the inlet. Then water flows into the tank before approaching the spillway. The segregated solver was used because it is multiphase flow with 2 materials, water and air, each with different velocity. The VOF and/or MMF models were used to deal with the multiphase fluids.

2.1 The volume of fluid model (VOF)

The VOF model, which was completely reported by Hirt and Nichols [12], is based on a concept of a fractional volume and the fact that the phases are not interpenetrating. The fields for all variables and properties are shared by the phases and represent volume-averaged values, as long as the volume fraction of each of the phases is known at each location. Thus the variables and properties in any given cell are either purely representative of one of the phases, or representative of a mixture of the phases, depending upon the volume fraction values. The standard interpolation schemes are used to obtain the face fluxes when a cell is completely filled with one phase while the geometric reconstruction scheme is used when the cell is near the interface between two phases. In each control volume, it must be filled with either a single fluid phase or a combination of phases while the volume fractions of all phases sum to unity. The same set of governing equations describing momentum and mass in a single-phase flow is solved throughout the domain, and the resulting velocity field is shared among the phases. The conservation equations, (1) and (2), are dependent on the volume fractions of all phases through the properties of density, ρ and dynamic viscosity, μ. The velocity in x_i and x_j directions are defined by u_i and u_j, respectively. The pressure, P, time, t, turbulent dynamic viscosity, μ_t, are also defined in these equations.

$$\frac{\partial \rho}{\partial t}+\frac{\partial \rho u_i}{\partial x_i}=0 \qquad (1)$$

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

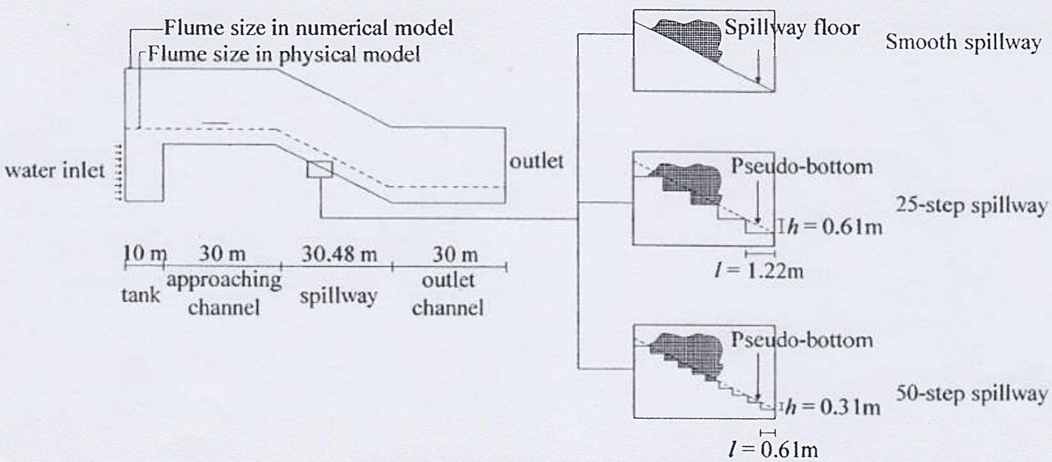

Fig. 1. Schematic diagram of the stepped spillway

2.2 The mixture multiphase flow model (MMF)

The MMF is a simplified multiphase model that can be used where the phases move at different velocities, but assume local equilibrium over short spatial length scales. The mixture model can model n phases by the conservetion equations for the mixture and the volume fraction equation for the secondary phases, as well as algebraic expressions for the relative velocities. The continuity equation for the mixture, (3), is defined by velocity of mixture, $\bar{\upsilon}_m = \frac{\sum_{k=1}^{n}\alpha_k \rho_k \bar{\upsilon}_k}{\rho_m}$, and density of mixture, $\rho_m = \sum_{k=1}^{n}\alpha_k\rho_k$. The fraction, density, and velocity of each phase can be defined by α_k, ρ_k, and $\bar{\upsilon}_k$, respectively.

$$\frac{\partial}{\partial t}(\rho_m)+\nabla\cdot(\rho_m\bar{\upsilon}_m)=0 \qquad (3)$$

The momentum equation for the mixture, (4), can be obtained by summing the individual momentum equations for all phases. The dynamic viscosity, $\mu_m = \sum_{k=1}^{n} \alpha_k \mu_k$, and the velocity difference between each phase and mixture, $\bar{\upsilon}_{dr,k} = \bar{\upsilon}_k - \bar{\upsilon}_m$, are also defined.

$$\frac{\partial}{\partial t}(\rho_m \bar{\upsilon}_m) + \nabla \cdot (\rho_m \bar{\upsilon}_m \bar{\upsilon}_m) = -\nabla P + \nabla \cdot \left[\mu_m \left(\nabla \bar{\upsilon}_m + \nabla \bar{\upsilon}_m^T\right)\right] + \rho_m \bar{g} + \nabla \cdot \left(\sum_{k=1}^{n} \alpha_k \rho_k \bar{\upsilon}_{dr,k} \bar{\upsilon}_{dr,k}\right) \tag{4}$$

The differences between the models are the manner in which they handle phase interpenetration and the phase velocities. With these two differences, the initial boundary condition must be different. The air velocity in mixture model should be set at zero and then reduced to homogeneous multiphase model while the air velocity in VOF model should be the same as water velocity. Flow over different kinds of spillways produce different patterns and have different effects.

For operating conditions, the specified operating density, 1.225 kg/m^3, was used with gravitational acceleration, -9.81 m/s^2, and operating pressure 101,325 Pa. The boundary conditions were set by using water velocity at water inlet. The Realizable k-ε model [13], which is a relatively recent development from the standard k-ε model, was used to simulate turbulence. The modeled transport equations for turbulent kinetic energy, k, and turbulent dissipation rate, ε, in the realizable k-ε model are (5) and (6), respectively. The generation of k due to the fluid shear, $G_k = \mu_t S^2$, generation of k due to buoyancy, G_b, effect of compressibility on turbulence, Y_M ,source terms of kinetic energy, S_k, $C_1 = \max\left[0.43, \frac{\eta}{\eta+5}\right]$, $\eta = S\frac{k}{\varepsilon}$, $S = \sqrt{2S_{ij}S_{ij}}$, source terms of dissipation rate, S_ε, C_2 = 1.9, kinematic viscosity, v, $C_{1\varepsilon}$ = 1.44, relation of flow velocity in x_i and x_j -direction, $C_{3\varepsilon} = \tanh\left|\frac{v}{u}\right|$ are defined in both equations.

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon - Y_M + S_k \tag{5}$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_j}(\rho \varepsilon u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \rho C_1 S_\varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\upsilon\varepsilon}} + C_{1\varepsilon}\frac{\varepsilon}{k} C_{3\varepsilon} G_b + S_\varepsilon \tag{6}$$

3. Physical Model

The experiments were obtained using an outdoor testing facility located at Colorado State University Engineering Research Center tested by Ward [14]. The facility permits large scale experiments so the water was supplied from nearby Horsetooth Reservoir. The concrete chute is approximately 34.14 m long, 1.22 m wide, and 1.52 m deep on a 2:1 (horizontal: vertical) slope and has a total height of 15.24 m. Plexiglas windows with the size of 1.22 m by 1.22 m were installed at five locations in the dividing wall to provide observation of flow.

The spillways can be divided into 3 groups; smooth, 25 and 50 steps. The 25-step spillway height, h, is 0.61 m and length, l, is 1.22 m. The data on the 4th, 8th, 12th, 16th, and 20th steps were measured, perpendicular to the pseudo-bottom. The 50-step spillway height, h, is 0.31 m and length, l, is 0.61 m. The data on the 7th, 15th, 23th, 31th, and 39th steps were measured, perpendicular to the pseudo-bottom. The Characteristics of flow over spillways are shown in Table 1.

Air concentration and velocity instrumentation were mounted on a point gage and carriage system. All profiles were taken along the centerline of the flume. The lowest points were taken at approximately 0.015 m from the tip of the step. The highest points were taken where both instruments measured data that was near the dry-air readings and visually appeared almost out of the flow. Videotape recording and photographs were used to collect the flow pattern at the overtopping crest and along the spillway. Flow condition in this study may be described as high-velocity, turbulent, two-phase flow. The probe is sturdy and provides a means of continuous back flushing to ensure a single density fluid within the Pitot tube. Velocity from the back-flushing Pitot tube is determined by the difference in pressures at the kinetic and static ports while continuously back flushing to prevent air bubbles from entering the instrument. Therefore, a balance between ensuring that air does not enter the Pitot tube and the sensitivity of the pressure difference must be found.

Table 1. Characteristics of flow over spillways

Parameters	Smooth spillway				25-step spillway					50-step spillway				
	$S_{0.57}$	$S_{1.13}$	$S_{1.70}$	$S_{2.27}$	$T_{0.57}$	$T_{1.13}$	$T_{1.70}$	$T_{2.27}$	$T_{3.28}$	$F_{0.20}$	$F_{0.60}$	$F_{1.16}$	$F_{1.70}$	$F_{2.27}$
Discharge (m^3/s)	0.57	1.13	1.70	2.27	0.57	1.13	1.70	2.27	3.28	0.20	0.60	1.16	1.70	2.27
Critical depth (m)	0.28	0.45	0.58	0.71	0.28	0.45	0.58	0.71	0.91	0.14	0.29	0.45	0.58	0.71
y_c/h	-	-	-	-	0.46	0.73	0.96	1.16	1.48	0.46	0.96	1.48	1.91	2.32
Flow regime	-	-	-	-	NA	NA	SK	SK	SK	NA	SK	SK	SK	SK

Remarks: NA is Nappe flow, SK is Skimming flow

4. Results and Significances

4.1 Flow characteristics on step

For the flow regime, Chinnarasri and Wongwises [15] proposed the minimum critical flow depth required for the onset of skimming flow on horizontal and inclined steps for $0.10 \le \frac{h}{l} \le 1.73$, with θ = angle of the upward inclined step in degrees, is

$$\frac{y_c}{h} = (0.844 + 0.003\theta)\left(\frac{h}{l}\right)^{-0.153+0.004\theta} \tag{7}$$

The maximum critical flow depth for the nappe flow regime is

$$\frac{y_c}{h} = 0.927 - 0.005\theta - 0.388\left(\frac{h}{l}\right) \tag{8}$$

For the present study, at 0.57 m^3/s and $\frac{y_c}{h} = 0.46$, nappe flow existed with ponded water in the interior of the step beneath a cascading free jet. At 1.13 m^3/s, with $\frac{y_c}{h} = 0.73$, partial impact of the flow near the end of step and incomplete filling of the step cavity suggest a partial nappe flow regime. The condition of skimming flow was first observed at 0.37 m^3/s or greater. Therefore, the flow regime at discharges of 2.83 and 3.28 m^3/s, which $\frac{y_c}{h} = 1.34$ and 1.48, respectively, were skimming flow. The complete submergence of the steps with water flowing down the slope as a coherent stream cushioned by recirculating vortices in the interior of the step was found. However, the general flow conditions were extremely turbulent along the entire spillway with erratic flow patterns and significant splash occurring at all flow rates.

The flow direction, and location of the recirculating vortices on the step from the simulation both from VOF and MMF models are similar. There are two zones; lower and upper zones, along the spillway. Closing to the step is lower zone with the recirculating vortices rotates clockwise and is located in the triangular zone of the step corner. The upper zone is far from the step and drops of water flow through the air. The results for the VOF model calculations can separate the air from the water flow so the connected surface between air and water can be seen clearly and the flow direction of water flow can be shown.

4.2 Velocity profiles

For the velocity profiles tend to have the same shape beginning with velocity gradually increasing from the bed until a maximum velocity gradient is reached. At some point in the upper region of the depth, an immediate change is observed where the velocity abruptly increases or decreases. Velocity profiles similar to this shape have been observed in several studies of stepped spillway flow Flow conditions in the upper region consisted of a highly irregular, wavy surface above which large particles of water ejected from the main flow. It was hypothesized that shear stress develops from increased resistance on these particles due to atmospheric drag and the change in momentum and the return of particles back to the main flow results in a loss of velocity.

For both nappe flow and skimming flow, the VOF model shows better agreement with measurements compared to the MMF model. The velocity profiles for the discharges of 0.57 m^3/s, nappe flow, and 3.28 m^3/s, and skimming flow on the 16[th] step are shown in the Fig. 2.

A comparison of the velocity results from the numerical model and experiments shows that with a nappe flow regime, the maximum error in their values mainly amount to 30% in the zone upper when 0.10 m from the floor bed of the spillway. For a skimming flow regime lower than 0.20m, the error is quite high as well as the one from nappe flow while the maximum error is not more than 27% in the upper zone. Given the grid size of 0.1×0.1 m^2 for the numerical model with the finite volume method, the maximum error is from the point that is located on the center of the cell, at 0.05 m. The results of the lower zone, therefore, are interpolated from the boundary of the cell. With this reason, the error from the floor to the depth 0.20 m is quite high.

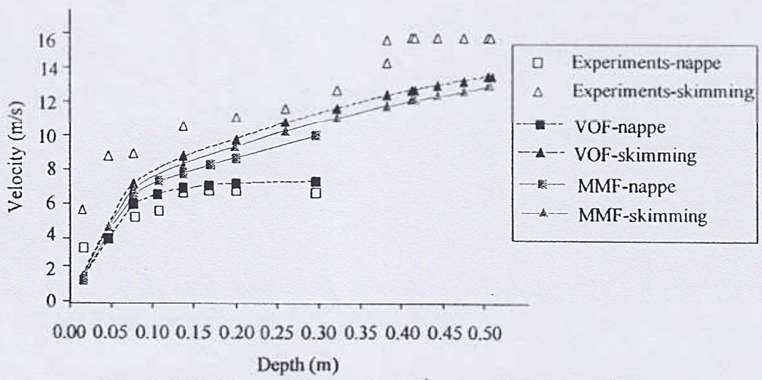

Fig. 2. Velocity profiles on the 16[th] step of 25-step spillway

4.3 Energy dissipation

The energy dissipation can be observed and calculated from the energy between the inlet section at the approach channel of spillway, E_0, and any section downstream, E_i. From many previous studies, the energy dissipation, $\frac{E_L}{E_0}$, is one of the dimensionless parameter which is widely used. The energy loss, E_L, is the difference between energy at the inlet section, E_0, and energy at any section, E_i. It can be written as $E_L = E_0 - E_i$.

To consider on the inlet discharge on stepped and smooth spillways, Fig. 3. shows the energy dissipation at the last station for both step and smooth spillways. The last observed station for smooth spillway is 25.76 m apart from the inlet and the 20[th] and 39[th] step for 25 and 50-step spillway, respectively. The relative length, which is the ratio between the distances from inlet to station over the spillway length, for all of those stations is 0.75. It is shown that for the same spillway, the energy dissipation decreases when the discharge, q, increases. The energy dissipation for the smooth spillway is rapidly decreased with increasing of discharge while the one for stepped spillway is gradually decreased. It can be stated that the stepped spillway is better used for higher design discharge than the smooth one because more energy can be dissipated due to the macro roughness of the steps.

Consideration on the certain discharge, the results indicate that a great amount of energy dissipation is occurred in a stepped spillway. It is also shown that for a given height, the energy dissipation increases when the number of steps increases. The results can be compared with the study from Rad and Teimouri [16] and the same trend is found as shown in Fig. 4. The spillway slope [16] was 26.6°, the number of steps was 32, and the Reynolds number was 10^5. The trend of results by Rad and Teimouri (2010) which $0.04 \le \frac{y_c}{ih} \le 0.13$ fits very well with the results from the present study which $0.02 \le \frac{y_c}{ih} \le 0.60$. Because each step acts as a macro roughness, the more steps can cause the thickness of turbulent boundary layer and more flow resistance and also significantly causes more energy dissipation. However, with high roughness, Chanson [3] reported the skimming flow will become fully developed and the stepped spillway behaves like a smooth spillway.

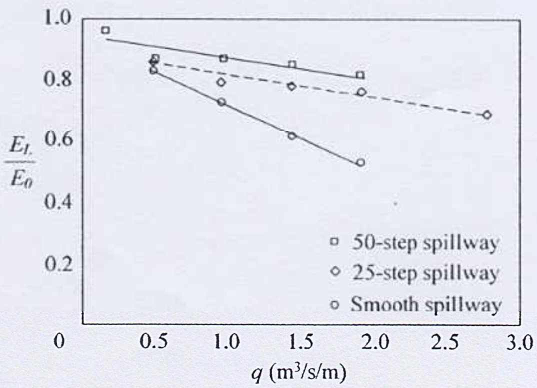

Fig. 3. Energy dissipation near the outlet

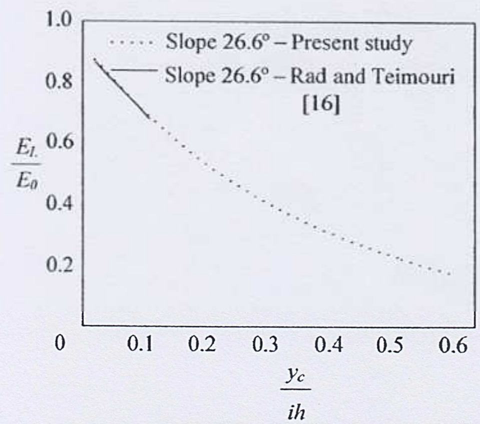

Fig. 4. Comparison of energy dissipation with previous studies

5. Conclusions

A numerical model using different multiphase flow models, VOF and MMF model, is used to study and compare the flow over a stepped spillway. The data from large-scale experiments are used to calibrate and verify the model. The grid size of quadrilateral meshes of 0.1×0.1 m^2 were used in the numerical model. For nappe flow, there is only an air pocket which is not clearly seen from the model. According to the simulation results for skimming flow, it is obvious that there is a recirculating vortex in the corner of the step, which is called lower zone, verified by the measurements. The upper zone is a wavy water surface in which air is trapped in the surface. The point of inception can be found in MMF model but without recirculating vortices shown on the step.

For the velocity profiles, they tend to have the same shape beginning with velocity gradually increasing from the bed until a maximum velocity gradient is reached. For both nappe flow and skimming flow, the VOF model shows better agreement compared to the MMF model. The maximum error in their values was 30% in the zone greater than 0.10 m from the floor bed of the spillway. For a skimming flow regime, the maximum error is not more than 27% in the upper zone. The energy dissipation from stepped spillways is compared with the smooth spillway. It is also compared with the previous studies and the same trend was found.

6. Acknowledgments

The authors thank Professors James F Ruff and Robert N Meroney from Colorado State University for their supporting data, facilities and recommendations.

7. References

[1] Gonzalez, C.A. and Chanson, H. 2006. Air entrainment and energy dissipation on embankment spillways. Proceedings of the International Symposium on Hydraulic Structures, IAHR Symposium, 12-13 October, Ciudad Guayana, Venezuela.

[2] Tabbara, M., Chatila, J. and Awwad, R. 2005. Computational simulation of flow over stepped spillways. Computers and Structures, Vol.83, pp.2215–2224.

[3] Chanson, H. 1994. Hydraulics of skimming flows over stepped channels and spillways. Journal of Hydraulic Research, Vol. 32, No.3, pp.445-460.

[4] Qian Z.D., Hu, X.Q., Huai, W.X. and Amador, A. 2009. Numerical simulation and analysis of water flow over stepped spillways. Science in China Series E:Technological Sciences, Vol.52, No.7. pp.1958-1965.

[5] Benmamar, S., Kettab, A. and Thirriot, C. 2003. Numerical simulation of turbulent flow upstream of the inception point in a stepped channel. Proceedings of 30th IAHR Congress, 24-29 August, Thessaloniki, Greece.

[6] Dong, Z.Y. and Lee, J.H. 2006. Numerical simulation of skimming flow over mild stepped channel. Journal of Hydrodynamics Series B, Vol.18, No.3, pp.367-371.

[7] 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, No.7, pp.683-688.

[8] Henderson, F.M. 1966. Open channel flow, Macmillan, New York.

[9] Boes, R. and Hager, W.H., 2003. Two-phase characteristics of stepped spillways. Journal of Hydraulic Engineering, Vol.129, No.9, pp.661-670.

[10] Gonzalez, C.A. and Chanson, H., 2004. Scale Effects in Moderate Slope Stepped Spillways. Experimental Studies in Air-Water Flows. In: Hubert Chanson and John Macintosh, 8th National Conference on Hydraulics in Water Engineering, 13-16 July, Australia.

[11] Chanson, H. and Gonzalez, C.A., 2005. Physical Modelling and Scale Effects of Air-Water Flows on Stepped Spillways. Journal of Zhejiang University, Vol.6A, No.3, pp.243-250.

[12] Hirt, C.W. and Nichols, B.D., 1981. Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries. Journal of Computational Physics, Vol.39, No.201, pp.201-225.

[13] Shih, T.H., Liou, W.W., Shabbir, A., Yang, Z. and Zhu, J., 1995. A new k-ε eddy viscosity model for high Reynolds number turbulent flows. Computers and Fluids, Vol.24, No.3, pp.227-238.

[14] Ward, J.P., 2002. Hydraulic design of stepped spillways, Dissertation for the degree of doctor philosophy, Colorado State University, 245pages.

[15] Chinnarasri, C. & Wongwises, S. 2004. Flow regimes and energy loss on chutes with upward inclined steps. Canadian Journal of Civil Engineering, Vol.31, No.5, pp.870–879.

[16] Rad, I.N. and Teimouri, M., 2010. An investigation of flow energy dissipation in simple stepped spillways by numerical model. European Journal of Scientific Research , Vol.47, No.4, pp.544-553.