Analysis of Pressure Profiles and Energy Dissipation across Stepped Spillways having Curved Treads using Computational Fluid Dynamics

Najam us Saqib*[1], Kamran Ansari[1], Munir Babar[1]
[1]US Pakistan Centre for Advanced Studies in Water, Mehran University of Engineering and Technology Jamshoro, Pakistan
*Corresponding author email: -najam.fraz@gmail.com

Abstract
Stepped spillways are commonly used as energy dissipaters in hydraulic structures. The superiority of stepped spillways over others is due to their effective energy dissipation and better aeration. The magnitude of energy dissipation depends upon the geometry of the steps and water flow rate. Alteration in step geometry can enhance energy dissipation. In this study, curved tread stepped spillways were modelled. The sensitivity of energy dissipation and pressure distribution to the angle of suspension were tested to check for their improvement. Four fourteen-steps stepped spillways with slope 1:0.84 were modelled using Froude's number non-dimensional similarity. The treads of steps were made curved at three angles of suspensions i.e., 30º, 60º, and 90º. These models were tested for a low unit-discharge. The simulations were performed by FLOW 3D® software and using Renormalization Group (RNG) turbulence model for a flow rate of 0.027 m^3/s followed by the model calibration. The 3D Reynolds-averaged Navier–Stokes equations were solved, which included sub-grid models for air entrainment, density evaluation, and drift-flux, to capture free-surface flow over the stepped spillway. Although curved tread may save significant material cost, yet the proximity of negative pressures increases along the vertical surface of curved tread steps, as compared to the simply stepped spillway. Moreover, no significant improvement in energy dissipation was observed.

Keywords: - Stepped Spillways, FLOW 3D®, Computational Fluid dynamics, Hydraulic Structures, Pressure Profiles

1.Introduction

Spillways are the integrated part of the dam as they allow safe passage of overtopping flow [1]. Stepped spillways contains steps that induces the macro roughness and projects the high turbulence in flow [2]. Geometry of steps is very important to determine the flow parameters over the steps and at downstream [3-4] . Energy dissipation to reduce the length of stilling basin and promote the safe flow over the stepped spillway, and pressure variations over the steps to study the aeration patterns, are the most significant flow parameters that many researchers investigated [5-6]. Previously researchers adopted the physical modelling to get the insight knowledge about the stepped spillway. For example Baylar et al.[7] performed model studies to check the aeration efficiency of stepped spillways, in particular the effects of varying chute angle and step height. Juny et al.[8] did the model studies to develop the pressure profiles on the horizontal and vertical surfaces of the steps in the skimming flow region, in order to develop the result for duration of negative pressures. Amador et al.[9] did the model studies on steeply sloped spillways, with high velocity flows, to investigate cavitation damage, along with formulation of mathematical expression to calculate the location of inception point. It was found that vertical surfaces of steps are prone to cavitation. Gonzalez and Chanson [10] did experimental study on stepped spillway, using Froude number similitude in large size experimental facilities, using 10 configurations which included smooth and inclined steps, and steps with devices to enhance the energy dissipation. It was found that inclined steps can enhance the energy dissipation up to 20%.Felder and Chanson [11] did the model studies to look for energy dissipation due to the production of turbulence due to steps. It was concluded that energy dissipation during the skimming flow is maximum for low discharge as compared high flow rates. Felder and Chanson [5] investigated the stepped spillways with non-uniform heights and found that there is not improvement in energy dissipation instead there is more turbulence for smaller flow rates. Chamani et al.[12] Investigated the model studies on stepped spillway with stones and gabions and found that energy dissipation as improved as compared to horizontal steps. Hamedi et al. [13] did model studies to study impact of inclined steps and inclined steps with end sills on energy dissipation. It was found that energy dissipation has considerably increased with the use of end sills in inclined steps. With the improvement in computer technologies many researchers employed numerical techniques to get the use results about the stepped spillways. Yasuda et al. [14] used k-e turbulence model to study a model stepped spillway using FLUENT software to make pressure profiles on the horizontal and vertical surface of the steps. Tabbara et al. [15] used ADINA software to simulate the turbulence flow over the stepped spillways of various step configurations. Salmasi & Özger [16] used neuro fuzzy approach to simulate the stepped spillways with different slopes and steps. They found that neuro fuzzy approach is best for the case calculation of energy dissipation as compared to regression analysis. Mansoori et al.[17] used FLOW $3D^{®}$ to evaluate the step geometry, in which steps contained Λ-shaped steps at angel of 25 degree in terms of calculating the energy dissipation in the steps spillways. Mohammad et al.[1] used FLOW $3D^{®}$ to check the effects on flow rate, chute slope and number of steps on energy dissipation. It was found that as flow rate increases, energy dissipation increases. On the other hand, if number of steps are increased, the energy dissipation is increased. The present study also employs a numerical method to look for effect of curving the tread of a spillway model on the energy dissipation and pressure profiles. Latest CFD (Computational fluid dynamics) commercial code FLOW $3D^{®}$ was used to carry out this study. To perform the calibration results from FLOW 3D were compared with previously published experimental work.

2.Materials and Methods

To carry out the confidence in the study, a calibration process is needed. For this purpose, partial experimental results were taken and compared with the same model results from FLOW $3D^{®}$. Later a model was selected using dimensional analysis and its steps were made curved by three angles of suspension i.e., 30 degrees,60 degrees and 90 degrees.

2.1 Model Calibration

In order to develop the confidence over the CFD (Computational Fluid Dynamics) software package FLOW $3D^{®}$, experimental results for horizontal and vertical pressure profiles at the step no 13 of physical model adopted by [6] were accounted. The model comprised of 13 steps and had a slope of 1 V: 0. 75H.The spillway model had a design head of 9.7 cm and the equation 1 WES (Water Ways Experiment station) was used to carry out the curve for the crest.

$y = 3.632x^{1.85}$ (1)

Out of the 13 steps, first 5 steps were transitional steps, while remaining steps have uniform dimensions. Width of spillway model was 30 cm and it had a 3 m long approach channel on the upstream side. There was no outlet control at tail water channel. The test discharge of the spillway was 0.02 m^3/sec which was measured by the triangular weir installed before the approach channel. Optical fiber laser Doppler anemometer produced by DANTEC Company Denmark was used to measure the velocities at different steps. While on the other hand five piezometers tube were fixed to measure the pressure at the horizontal and vertical surfaces of the steps.

2.2 Mathematical Model and Simulation Setup

The commercial code FLOW $3D^{®}$ was used to perform this study. It can solve 3D Reynolds-averaged Navier-Stokes (RANS) equations for one fluid including RNG k-ε turbulence model numerically [18] . FLOW $3D^{®}$ uses TurVOF method [19] for interface tracking.

2.3 Mass Continuity Equation

As the flow over the stepped spillway is two phase flow (air water flow) therefore continuity equation in the form of volume weighted average density and velocity of corresponding phases (air and water), is given as follows

$$\frac{\partial \rho_m}{\partial t} + \nabla \times (\rho_m \boldsymbol{u_m}) - \nabla \times (\vartheta \nabla \rho_m) = 0 \quad (2)$$

$$\vartheta = \frac{S_c u_m}{\rho_m} \quad (3)$$

Where ρ_m and $\boldsymbol{u_m}$ are average density and velocity respectively. The term $\nabla \times (\vartheta \nabla \rho_m)$ is the turbulent diffusion term which only makes sense for turbulent mixing process in fluids with non-uniform density. On the other hand, S_c is the constant that is equal to reciprocal of turbulent Schmidt number and u_m is volume weighted average dynamic viscosity.

2.4 Momentum Equation

Momentum equation for the fluid mixture is given below

$$\frac{\partial \rho_m \boldsymbol{u_m}}{\partial t} + \nabla \times (\rho_m \boldsymbol{u_m} \boldsymbol{u_m}) = -\nabla P + \rho_m \boldsymbol{g} + \nabla \times \tau \quad (4)$$

Where τ is Reynolds's stress tensor, P is pressure and $\boldsymbol{g}$ is gravitational acceleration.

2.5 RNG k- ε Turbulence Model

In this study RNG k-ε Turbulence Model which is capable of accounting the smaller scales of motion was selected The governing equations are given below

$$\frac{\boldsymbol{D}}{\boldsymbol{Dt}}(\rho_m k) = \nabla \times (\rho_m D_k \nabla k) + P_k - \rho_m\, \varepsilon \quad (6)$$

$$\frac{D}{Dt}(\rho_m k) = \nabla \times (\rho_m D_\varepsilon \nabla\, \varepsilon) + \frac{C_1 P_k \varepsilon}{k} - \frac{C_2 \rho_m \varepsilon^2}{k} \quad (7)$$

Where D_k and D_ε shows the effective diffusivity of k and ε. P_k is the generation k due mean velocity gradients. C_1 is equal to 1,42 and C_2 can be calculated from C_2=1.68k.

2.6 VOF Model

FLOW 3D® uses TruVOF technique to track the interface between two non-soluble fluid (Air and water in our study). It uses an indicator scalar, whose value range from 0 to 1 representing fractional volume of main fluid i.e., is water in our case. TruVOF also applies suitable boundary condition at the fluid interface i.e., atmospheric pressure in our case. The equation for f is given below

$$\frac{\partial f}{\partial t} + \boldsymbol{u_m} \times \nabla \times f = \nabla \times (\vartheta \nabla f) \quad (8)$$

The term on right hand side stands for turbulent diffusion.

2.7 Air Entrainment Model

Air Entrainment Model in FLOW 3D® is based on the assumption that air Entrainment in the free surface will occur. This will occur due to instabilities created by turbulence produced by flow over the complex geometry. Thus, this turbulence overcome the stabilizing forces, caused by gravity and surface tension. As a result, air with the volume δV may be entrained into the fluid, which can be expressed by the following equations.

$$L_T = \frac{CNU^{\frac{3}{4}} k^{\frac{3}{2}}}{\varepsilon_T} \quad (9)$$

$$P_t = \rho_w k \; ; \; P_d = \rho_w g_n L_T + \frac{\sigma_{sur}}{L_T} \quad (10)$$

$$\delta V = \{k_{air} A_s \left[\frac{2(P_t - P_d)}{\rho_w}\right]^{\frac{1}{2}} \text{for } P_t > P_d \text{ ,0 for } P_t < P_d\} \quad (11)$$

Where CNU is a constant and has a value of 0.09, L_T is turbulence scale length and ε_T is turbulent dissipation. ρ_w is the water density σ_{sur} is the coefficient of surface tension. A_s is the surface area, k_{air} is proportionality constant. δV is the volume of air entrained per unit time.

2.8 Density Evaluation Model

As the air is entrained air, the density of water became non uniform. There is the formation of air water mixture who density can be computed as follows

$$\rho_m = (1 - C_a)\rho_w + C_a \; \rho_a \quad (12)$$

Here ρ_m is the mixture density while ρ_a is the air density. C_a is the air concentration.

2.9 Drift Flux Model

According to [20] phase drag and bubble diameter can be modeled using drift flux model. According to this model air particles are dispersed into air over the continuous water flow. The relative velocity between the velocity of dispersed particles and continuous water flow are considered steady. The air transport equation is given below

$$\left(\frac{1}{\rho_w} - \frac{1}{\rho_a}\right) \nabla P = \left(\frac{f\rho_w + (1-f)\rho_a}{(1-f)\rho_w \rho_a}\right) \times K u_r \quad (13)$$

K is cell drag coefficient is volume fraction of water, u_r shows relative or slip velocity. K can be computed from equation 14 given below

$$K_p = \frac{1}{2} A_p \rho_w \times \left(C_d U_r + \frac{12\mu_w}{\rho_w R_p} \right) \qquad (14)$$

$$K = \frac{(1-f)}{V_p} (K_p) \qquad (15)$$

K_p is single drag coefficient. A_p is the cross-sectional area of the bubble. U_r depends upon u_r, μ_w is dynamic viscosity of water, C_d is user defined drag coefficient and R_p denotes bubble diameter.

2.10 Simulation Setup

Geometry of the calibration model [6] was created in SOLIDWORKS. It was then imported to FLOW 3D® in the form of STL (Stereolithography) 3D file. Different sub model like air entrainment, drift flux, density evaluation etc. as discussed in section 2.3 to section 2.9 were selected. Boundary conditions were selected as pressure boundary with fluid elevation at the upstream, out flow boundary condition at the end stilling channel, wall boundary condition at the sides and at the bottom were selected. Pressure boundary condition was selected at the top corresponding to the atmospheric pressure.

2.11 Grid Sensitivity Analysis

Grid sensitivity analysis was performed at the step No 13 of the calibration model [6]. Four types of mesh size (20 mm,15 mm,10 mm, and 5 mm) were selected. From **Figure** 1 it is clear that mesh size of 5 mm is good to simulate the accurate results. Large errors were found by the use of 20 mm mesh size, while the least errors were yielded by the 5 mm mesh size. Although 5 mm mesh adopts more computational cost, but it is compromised with accuracy. Here the mesh size of 2 mm or 1 mm can be also be used but they need huge computational cost.

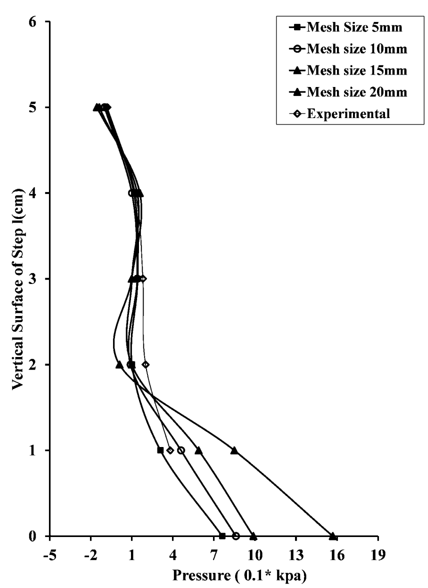

Fig. 1 Grid Sensitivity Analysis

2.12 Pressure Profile Calibration

To validate the model, the pressure profiles over the vertical and horizontal surface of the step no 13 of calibration model [6] were made and compared. It can be seen in Figure 2 & 3, that FLOW 3D® is accurate enough to encounter the complex flow over the stepped spillways. No negative pressure profile is seen in horizontal surface. While the vertical surface of the steps shows the negative pressure.

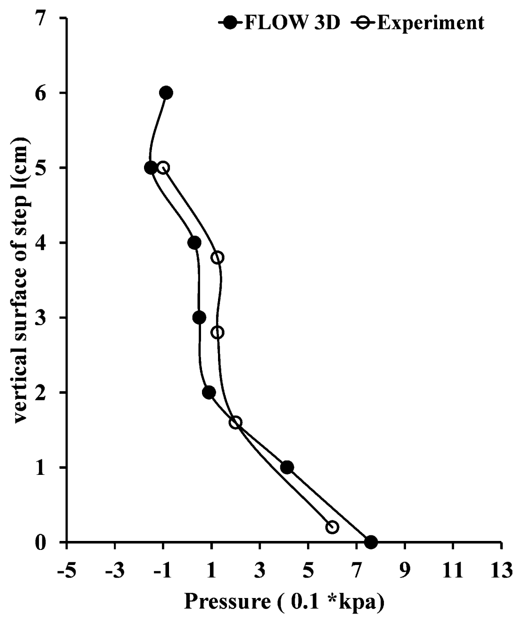

Fig. 2 FLOW 3D vs Experiment at vertical surface

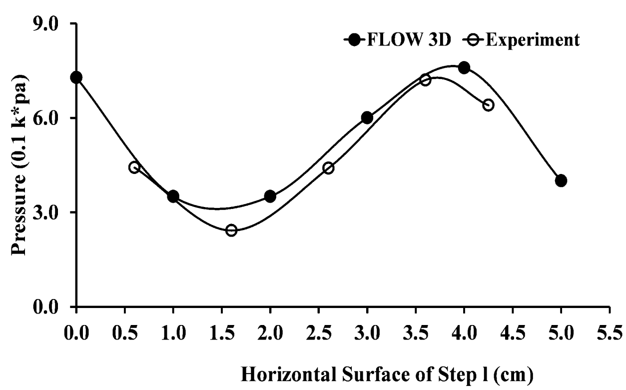

Fig. 3 FLOW 3D vs Experiment at horizontal surface

3.Spillways Models with Curved Treads

A model was selected using Froude's number analysis. To begin with, ogee spillways of Khanpur Dam Pakistan was considered as prototype. Khanpur dam is located about 50 km away from

Pakistan's Capital city Pakistan. It is built on Haro River. It forms a lake which provides the drinking water to Rawalpindi and Islamabad. General features of Khanpur dam are given below [21].

Location	Taxila on Haro River
Catchment Area	308 square miles
Design Flood discharge	166000 cusecs
Main Dam type	Earth and Rock fill
Maximum dam height	167 ft.
Dam crest length	1560 ft.
No of spillways	5

Using Froude's number analysis of model generation, a model was constructed in the ratio of 1:50. Head and number of steps were considered in a way to minimize the scale effects [22]. General features of model are presented below.

Type of spillway	Stepped
No of steps	14
Height of model	1.06 m
Maximum design head	213 mm
Maximum design discharge	0.064 m^3/s

Depending upon the design discharge, the crest equation was determined by USBR (United States Bureau of Reclamation) equations [23] based on WES(Water Ways Experiment station). A total of four equations for crest were selected with the vertical approach height. A head of 100 mm was considered over the spillway to mitigate the scaling effects [22] as well as 14 number of steps renders it in the skimming flow region. After that each tread of this step spillway were made curved by different angle of suspension that are 30°, 60° and 90°as presented in Figures **4** and **5**. Based on angle of suspensions at the treads each spillway model was given name as presented in table **1**.

Table 01:- Steps configurations tested

Simulation No.	Case	Spillway Model	Tread Type	Angle Of Suspension(º)
1	C0	Simple Stepped	Horizontal	0
2	C30	Simple Stepped	Curved	30
3	C60	Simple Stepped	Curved	60
4	C90	Simple Stepped	Curved	90

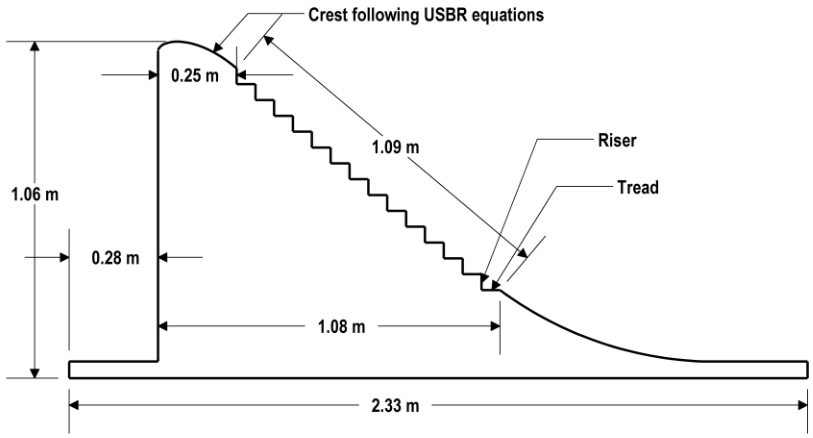

Fig.4 Step configuration for C0 stepped spillway model

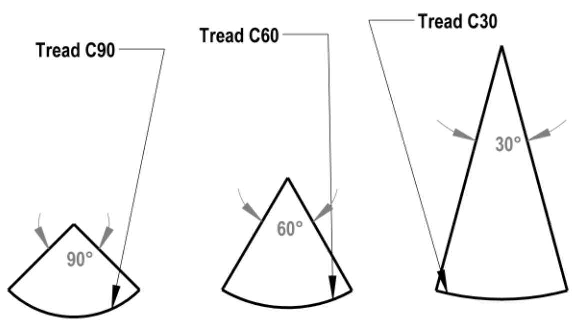

Fig.**05** Tread configurations for C30, C60 and C90

Simulation setup was initiated by creating the geometry in SOLIDWORKS. The geometry was imported to FLOW 3D® in the form of STL. As showed by grid analysis 5 mm mesh size was selected. A unit flow rate of 0.027 m^3/s was selected to pass over each of the spillway. Head at the inlet, symmetry at the sides, wall at the bottom, outflow at the outlet, atmospheric pressure at the top was selected as the boundary conditions. RNG k- ε Turbulence Model was selected to account for turbulence in flow. Different sub models VOF model, drift flux model, density evaluation model, air entrainment model as discussed in section 2.3 to section 2.9 were selected. Each Simulation was run till steady state condition. Pressure profiles along the horizontal and vertical surfaces were drawn on the treads (horizontal for C0 and curved for C30,60 and C90) and risers of the steps. Hydraulic heads were obtained from FLOW 3D® at the upstream and downstream of spillway model. Difference was calculated to get the percent energy dissipation.

4.Results and Discussions

4.1. Pressure Profiles Along the horizontal Surfaces

Figure 6 presents the pressure profiles for horizontal pressure surface of simple stepped spillways i.e., C0 tread configuration (Figure 4). Here x represents a particular point along the length and L is total length of tread. As seen the nature of pressure profiles is consistent with studies done by [13–15]. Pressure goes down at the start of the steps, and then it increases towards the tip of the steps. This is different for each step as each step exhibits different flow conditions. Near the tip of the step the pressure drastically decreases as the velocity along the step tip is very high due to a constant stream of water flow [25]. Figure 7 presents the pressure profiles along the curved surface of tread for C30 configuration. It can be seen that for C30 step No. 11 and step No. 1 experience the maximum pressures. Similarly, lowest pressure was seen at step No 12. This is different to simple stepped spillways, where step No 14 showed the highest pressures and step No 13 showed the lowest pressures just closer to vertical surface of the step.

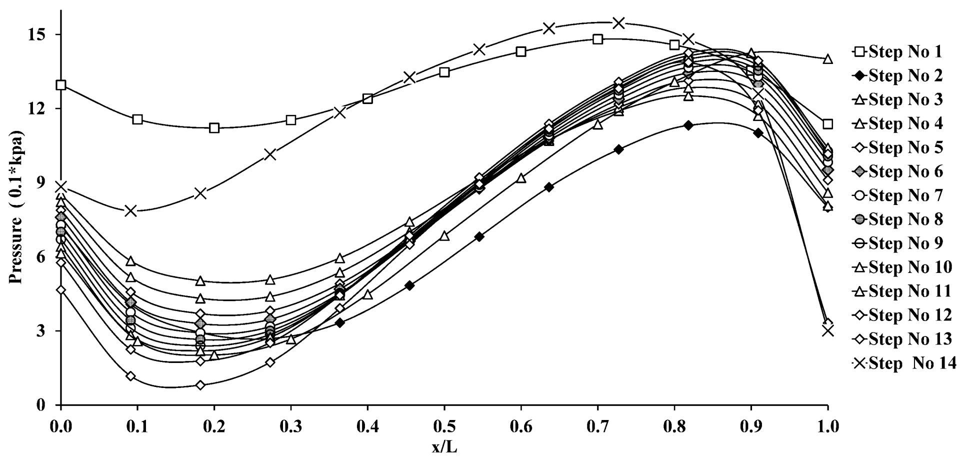

Fig.6 Pressure profile at the horizontal surface for C0 Step

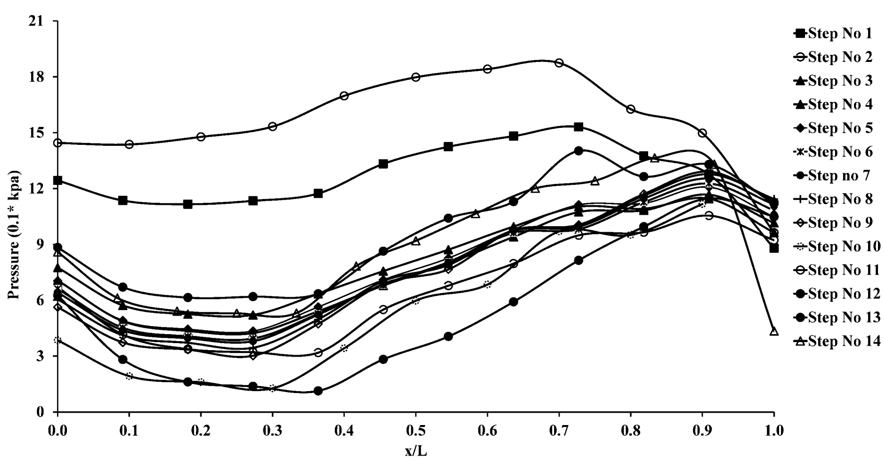

Fig.7 Pressure profiles at horizontal surface of C30

Figure 8 shows the pressure profiles over the curved tread of C60 configuration. It can be seen that the pressure is highest at the step No. 11 and lowest at step No. 2. Pressure drop at the end of tread is more as compared to C0 configuration. Figure 9 depicts the pressure profiles for C90 configuration. Same as C60 the pressure is highest at step No 11 and lowest at step no 12.

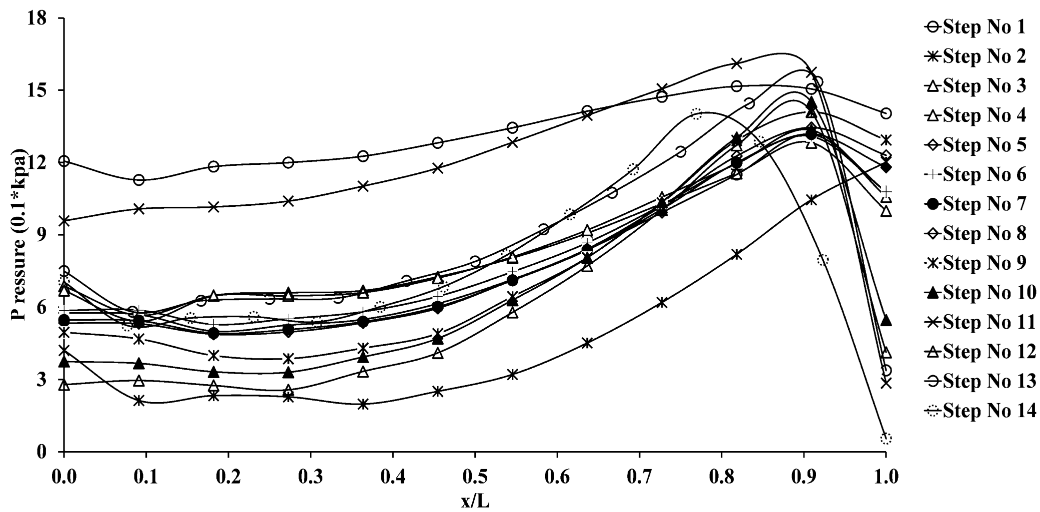

Fig.8 Pressure profile at the horizontal surface for C60 Step

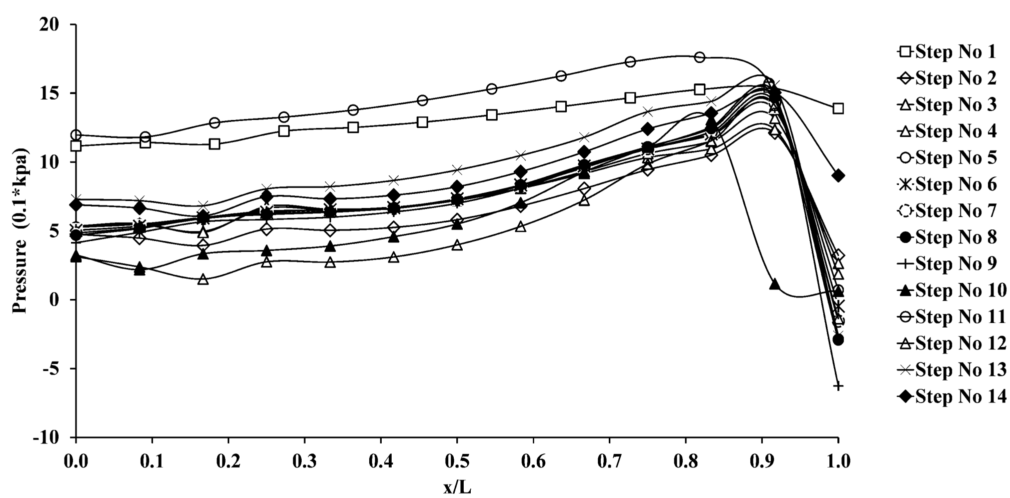

Fig.9 Pressure profiles at horizontal surface of C90

4.2 Pressure Profiles Along the Vertical/Curved Surfaces

When the pressure profiles were drawn at the vertical surfaces of all stepped spillway configurations, it was generally found that these surfaces of the steps are subjected to the negative pressure and can cause cavitation [26]. Moreover, due to angular structure of the tread, the values of negative pressures increase. **Figure 10** presents pressure profiles at the vertical surfaces, for risers of the steps for C0 stepped configuration. There is no change in the riser configuration, but as the tread is becoming curved, there is impact on nature of flow along the risers. It can be seen that maximum negative pressures are experienced by step No 14 and step No 2. The main reason behind this is the maximum velocities possessed by vortices formation at these steps. This changes when the tread (horizontal surfaces) are made curved. **Figrure 11** presents the pressure profiles at the risers of the steps for C30. It can be noted that step no. 12 shows the maximum negetive pressure that is equal to -2.8 kpa (kilopascals) which is greater as compared to simple stepped spillway where the maximum negetive pressure is -1.69 kpa.It can seen that C30 stepped spillway configration faced almost double the negative pressure as compared to simple stepped spillways.Therefore C30 is more vulnurable to cavitation damage as compared to simple stepped spillway. After step no 12,maximum negetive pressure is face by step no 4. As compared to simple stepped spillway,it can be seen that curvedness in tread is shifting negetive pressures at steps.

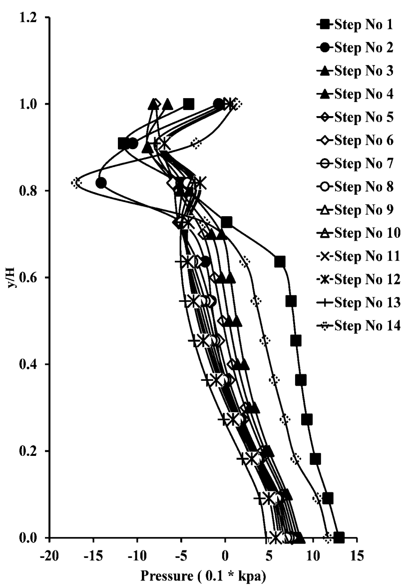

Fig.10 Pressure profile at risers for C0

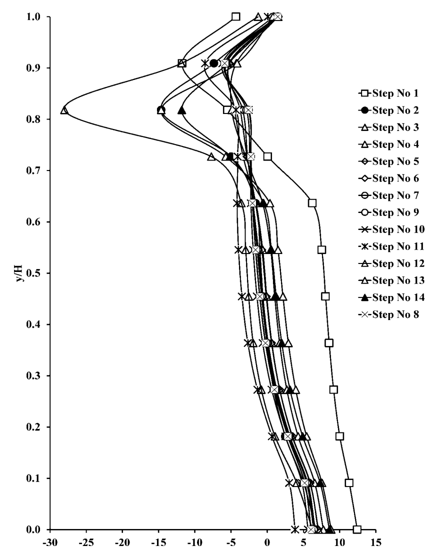

Fig.11 Pressure profiles at riser of C30

Figure 12 & 13 presents the pressure profiles for risers of C60 step configuration. As compared to simple stepped spillway, C60 configured stepped spillways also show the maximum negative pressure at step no 12 with the value of -2.1 kpa, which is less than maximum negative pressure face by C30. After the step no 12, the maximum negative pressure taken up was step no 5, as compared to C30 where it occurred at step no 4. Similarly, for C90 step configuration as shown in Figure 13 the maximum negative pressure is shown by step no 12 and step no 14 being the next. It shows the maximum negative pressure of -2.2 kpa, which is more as compared to simple stepped spillways. Moreover, it is less than C60 and C60 step configurations.

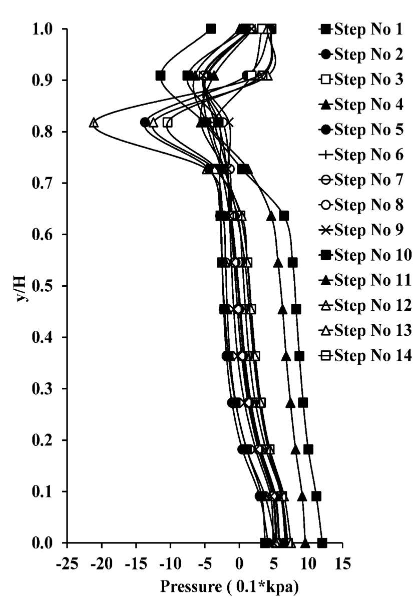

Fig.12 Pressure profile at risers for C60

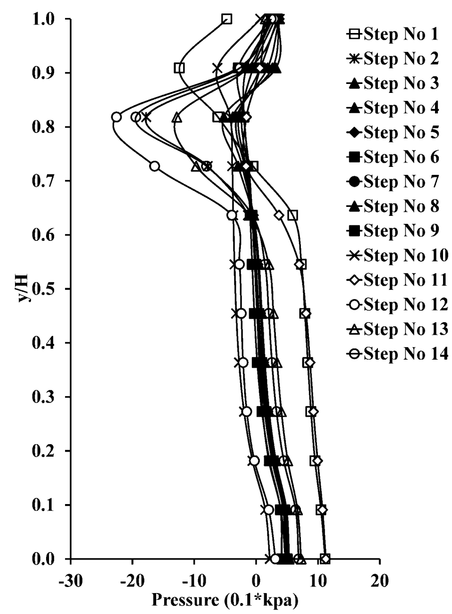

Fig.13 Pressure profiles at riser of C90

4.3 Energy Dissipation

Energy dissipation was calculated by the difference between head at upstream and the downstream stream heads. Moreover, as given in table 2, it can be seen that curving the tread at the flow rate of 0.027 m^3/s, there is no much considerable increment improvement in energy dissipation. Although curving the tread would save much of the material, in the prototype but yet the high negative pressure at the vertical surface may present the problems related to cavitation. Aerators can be used to sufficiently aerate the flow when the curved treads are needed to be used.

Table 2:- Energy dissipation

Type of Step	Difference in Heads at Upstream and Downstream(mm)	% Head Dissipation
C0	700.402	59.206
C30	709.577	59.981
C60	709.206	59.950
C90	708.222	59.867

5.Conslusion

The study briefly investigated the pressure variation and energy dissipation of a stepped spillway using latest computational fluid dynamics software FLOW3D®. This study is concluded as follows.

- Using Froude's number stepped spillway, a simple stepped spillway model was selected with 14 steps and slope of 1:0.8. USBR equations were employed to calculate the equation of the crest. A single flow rate of 0.027 m^3/s was selected, in an away to skim the flow over the steps. Later the treads of each step were made curved.

- At the flow rate of 0.027 m^3/s, pressure profiles along the horizontal and vertical surface of all the stepped spillways were drawn. It was found that as compared to simple stepped spillway, angled tread stepped spillway experience more negative pressure as compared to simple stepped spillway. It was also found that along the horizontal surface, the curved tread stepped spillway experience more pressure as compared to simple stepped spillway.
- Energy dissipation was found by calculating the head difference between the upstream and the downstream. 59 % energy dissipation was observed. However, with the curved stepped spillway no significant increment was found.

The scope of the study is limited as the numerical method of investigation is involved. The calibration was however performed, but still there is possible presence of error which can make the results of these study suspicions. The study is only limited to skimming flows and lower flow rates. The behavior may change when the curved stepped spillway may be subjected to higher flow rates or different slopes or number of steps. This can be performed in the future and more insight results can be obtained for curved tread stepped spillways.

6.Acknowledgments

Sincere thanks to USAID (United stated Agency for international development) for providing their support, data, and funds to complete this study.

7.List of Notations

x: Any point along horizontal /curved surface of step(cm)
y: Any point along vertical surface of step(cm)
L: length of tread(cm)
H: Height of the step(cm)
Fr: Froude's Number (Dimensional less)
ρ_m : Average Density(kg/m^3)
$\boldsymbol{u_m}$: Average Density (kg/m^3)
τ : Reynolds's stress tensor
P : Pressure(Kilopascals)
ϑ_{eff} : Effective kinematic viscosity(m^2s^{-1})
k : turbulent kinetic energy(J)
δV: Air entrainment model(mm^3)
L_T : Turbulence scale length(mm)
ε_T : Turbulent dissipation(mm)
ρ_w : Water density(kg/m^3)
ρ_a : Air density(kg/m^3)
C_a : Air concentration(mm^3)
K : cell drag coefficient
u_r: Relative or slip velocity(mm/sec)
K_p : Single drag coefficient
A_p : Cross sectional area of the bubble(mm^2)
μ_w : Dynamic viscosity of water (kg.ms)

8.References

[1]Tabari,Tavakoli,Effects of Stepped Spillway Geometry on Flow Pattern and Energy Dissipation,Arab. J.Sci.Eng, **41,**(4),1215–1224,(2016)
[2] Bung,Valero,Re-aeration on stepped spillways with special consideration of entrained and entrapped air, Geosci, **8,**(9) , 1–15 ,(2018).
[3] Hubert Chanson,Energy dissipation on stepped spillways, J. Hydraul. Eng,**121,**(1), 80–87, (1995).
[4] Tuna, Emiroglu,Effect of Step Geometry on Local Scour Downstream of Stepped Chutes, Arab. J. Sci. Eng,**38,**(3),3579–588, (2013).
[5] Stefan Felder,Hubert Chanson,Energy dissipation down a stepped spillway with nonuniform step heights. J. Hydraul. Eng. ASCE **137**(11), 1543–1548 (2011).
[6] Qun Chen,Dai,Liu, Volume of fluid model for turbulence numerical simulation of stepped spillway overflow,J. Hydraul. Eng. ASCE **128**(7), 683–688 (2002)
[7] Ahmet Baylar,Emin Emiroglu,Tamer Bagatur, An experimental investigation of aeration performance in stepped spillways, Water Environ. J, **20,**(1), 35–42, (2006).
[8] Juny Sánchez,Bladé,Joseph Dolz,Analysis of pressures on a stepped spillway, J. Hydraul. Res, **46,**(3), ,410–414,(2008)
[9] Amador,June Sánchez,Joseph Dolz, Developing flow region and pressure fluctuations on steeply sloping stepped spillways, J. Hydraul. Eng. ASCE, **135** ,(12), 1092–1100, (2009).
[10] Carlos Gonzalez,Hubert Chanson,Hydraulic Design of Stepped Spillways and Downstream Energy Dissipators for Embankment Dams.Dam Eng, **17,** (4), 223–244,(2007).
[11] Stefan Felder,Hubert Chanson,Energy Dissipation and Residual Energy on stepped spillways, 33rd IAHR Congr,1–8 ,(2009).
[12] Rajartnam Chamani,Characteristics of skimming flowover stepped spillways. J. Hydraul. Eng. ASCE, **125,**(4), 361–368, (1999).
[13] Amirmasod Hamedi,Abbas Mansoori, Ali Shamsai,A,Effects of End Sill and Step Slope on Stepped Spillway Energy Dissipation, J. Water Sci,Res.**12,**(6),1-2(2014).
[14] Youichi Yasuda,Masayuki Takahashi,Iwao Ohtsu, Discussion of 'volume of fluid model for turbulence numerical simulation of stepped spillway overflow, J. Hydraul. Eng. ASCE, **130,**(2),170–170,(2004).
[15] Mazen Tabbara,Jean Chatila,Ritta Awwad Computational simulation of flow over stepped spillways, J. Comput. Struct,**83,**(27), 2215–2224,(2005).
[16] Salmasi,F.; Özger,M.: Neuro-Fuzzy Approach for Estimating Energy Dissipation in Skimming Flow over Stepped Spillways.Arab. J. Sci. Eng.**39**(8), 6099–6108(2014).
[17] Abbas Mansoori,Shadi Erfanian,Farhad Khamchin Moghadam,A Study of the Conditions of Energy Dissipation in Stepped Spillways with Λ-shaped step Using FLOW-3D,Civ. Eng. J,**10**(3), 856-860(2017).
[18] Yakhot, V.;Orszag, S. A.: Renormalization group analysis of turbulence. I. Basic theory. Journal of scientific computing, **1**(1), 3-51(1986).
[19] Hirt, C.W.; Nichols, B.D.: Volume of fluid (VOF) method for the dynamics of free boundaries. JCP **39**, 201 (1981).
[20] Hirt,C.W.; Drift Model for Two-Component Flows.1–7,(2009).
[21] S. Ali,A.; Ahmed,S.I.; Technical Case Study & Design Of Gravity & Rockfill Dams.Bachelors Thesis,Department of Civil Engineering,FAST National University of Computer and Emerging Sciences,Pakistan (2015)
[22] Heller ,V.; Scale effects in physical hydraulic engineering models. J. Hydraul. Res. **49**(3),293–306(2011)
[23] J. C. Potter,C.J.;Box,P.O.: Waterways Experiment Station , Corps of Engineers US Army

Corps of Engineers.(1988)
[24] M. Sánchez,J.M.;Pomares,J.;Dolz,J.:Pressure field in skimming flow over a stepped spillway.*Proc. Int. Work.* Hydraul. Stepped Spillways.137 146(2000).
[25] Kositgittiwong,D.; Chinnarasri,C.; Julien,P.Y.: Numerical simulation of flow velocity profiles along a stepped spillway. Proc.Inst.Mech.Eng. Part E J.Process Mech. Eng. 227(4),327 335(2013).
[26] Yusuf,F.; Micovic,Z.; Prototype-Scale Investigation of Spillway Cavitation Damage and Numerical Modeling of Mitigation Options J. Hydraul. Eng.ASCE **146**(2), 1–12(2020)