ISSN No:-2456-2165

Hydraulic Simulation and Numerical Investigation of the Flow in the Stepped Spillway with the Help of FLOW 3D Software

Vihaan Sharma Jai
P P Savani University, School of Civil Engineering, India

Abstract:- How to transfer the flood of a dam to the downstream and reduce its energy is one of the most important issues in the design of a dam. In recent years, due to the energy loss along its length, the stepped spillways have received a lot of attention from researchers. In this research, the values of energy loss have been investigated with the help of flow 3D software with different turbulence equations and their comparison with laboratory values. The results obtained from this research show that the model of normalized groups or RNG has the highest level of accuracy among different turbulence models.

Keywords:- *Stepped Spillway, Energy Dissipation, Flow 3D Software.*

I. INTRODUCTION

Since the spillway of a dam is one of the most important structural components of any dam, the hydraulic knowledge of the flow in the spillways is considered one of the most important parts of the project. Flow along the structure and the reduction of the size of the stilling basin has been noticed more. A stepped spillway consists of a number of successive vertical aqueducts that continue from near the top of the crown to the stilling basin at the bottom. The flow of water on the structure can be Nappe flow or Skimming (non-Nappe flow). More Nappe flow occurs for low flow rates and large steps, and Skimming (non-Nappe flow) flow occurs for higher flow rates and a large number of steps. In falling flows, the flow characteristics change after passing a few steps, which means that there is a lot of air trapped in the toe of the step and strong vortices are created. It seems that the energy consumption is enhanced due to the transformation of momentum into the rotating fluid.

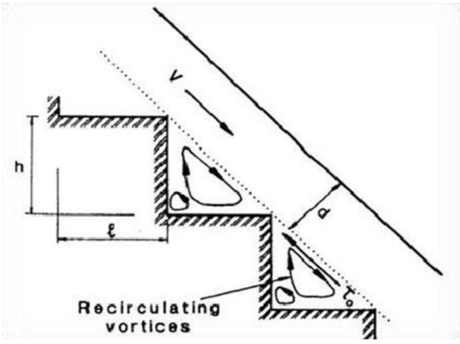

Fig. 1. Flow with a Nappe flow regime. *(figure caption)*

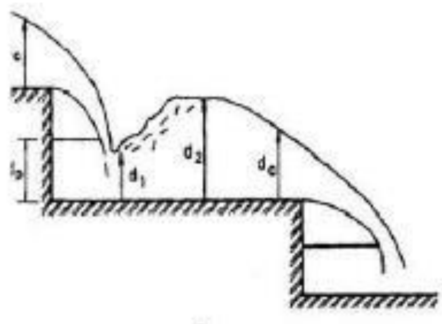

Fig. 2. Flow with a Skimming flow regime. *(figure caption)*

In Nappe flows, the characteristics of the flow change after several steps, which means that there is a lot of trapped air in the bottom of the step and violent eddies are created. It seems that the consumption of energy is strengthened due to the transformation of momentum into circulating fluid.

In free-surface turbulent flows, large amount of air may be entrapped and advected in the water current. The resulting air-water flows are frequently observed in natural water systems, where they are also relevant to water quality, ecological sustainability and integrated assessment within such systems[1]. Reeve et al performed an intensive set of tests with varying slope, stone size, and porosity were undertaken. The location of the inception points and the water depth at this point obtained from their investigation were compared with those from existing formulae. Two new empirical equations have been derived, on the basis of a regression analysis, to provide improved results for gabion stepped spillways[2]. Chatila et al used finite element computational fluid dynamics software, ADINA, to predict the free surface over an ogee spillway. The software predicted reasonable free surface results that are consistent with general flow characteristics over spillways[3]. An accurate description of the hydrodynamics in the non-aerated region of the skimming flow on stepped spillways is of outmost importance, particularly in small structures at large discharges[4]. Tebbara, Chatila and Awwad[5], Zhenwei, Zhiyan and Tao[6] investigated the flow pattern over the stepped spillways. The aim of this research is to simulate the flow field on a stepped spillway with the help of FLOW 3D software and compare the results of different turbulence models with each other and find the error values of each one. In other words, it has been assumed that different flow turbulence models give different answers to the same problem according to their solution methods, which indicates the difference in their calculation accuracy. As a result, by comparing the results obtained from each turbulence model with physical results, the accuracy and accuracy of each turbulence model can be calculated and finally the most accurate turbulence model can be introduced in flow simulation.

II. MATERIALS AND METHODS

A. Flow dominant equilibrium

Constancy and momentum equilibrium in uncompressed fluor flow is:

$$\frac{\partial p}{\partial t} + \frac{\partial p u_i}{\partial x} = 0 \quad (1)$$

$$\frac{\partial p u_i}{\partial t} + \frac{\partial}{\partial x_i}\left(p u_i u_j\right) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(\mu + \mu_1)\left(\frac{\partial u_i}{\partial j} + \frac{\partial u_j}{\partial i}\right)\right]$$
(2)

Where t is the time, Ui is the velocity parameter, xi is the axis parameter, p is the density, μ is the dynamic legit, is the turbulence legit, and p' is the modified pressure.

p' can also be calculated from:

$$P' = P + \frac{2\rho k}{3} \ (3)$$

Where p is the pressure and k is the motive kinetic energy. Also, k and equilibrium could be stated as:

$$\frac{\partial(\rho K)}{\partial t} + \frac{\partial(u_i k)}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right] + G + \rho\epsilon$$
(4)

$$\frac{\partial(\rho\epsilon)}{\partial t} + \frac{\partial(u_j \epsilon)}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_i}\right] + C_{1\epsilon}\frac{\epsilon}{k}G + C_{2\epsilon}\rho\frac{\epsilon^2}{k}$$
(5)

Where ϵ is Motive kinetic energy and k is motive legit that ϵ could be calculated by μ_t

As followed:

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

Where Cμ= 0.09 is the test standard. Prantel motive numbers for k and ε were included

C2ε=1.92and σ_ϵ=1.3 , σ_k=1.0 as the constants of ε equilibrium. Motive kinetic energy production G, based on the average velocity gradient is as followed:

$$G = \mu_t\left(\frac{\partial ui}{\partial x_j} + \frac{\partial uj}{\partial x_i}\right)\frac{\partial ui}{\partial x_j}$$
(7)

B. VOF

The VOF method is applied to several problems of environmental interest, including the dam-break problem, sluice gate operation, internal waves, and surge propagation in a dry channel [7]. VOF model is a monitoring tool of the free surface using Oleri constant meshes. In this model, an individual utility used from momentum equilibrium for the phases and the dense component of each phase was calculated in each calculative cell of the detection circle.

VOF method is based on the principle that two or more fluors could not penetrate into each other. For each fluor adding to the model, there was a variable called the ratio of the fluor density (α). The ratio of the fluor density was equal to the fluor density of a cell to the cell density. In each control density, the total ratio of the fluor density in all fluor stages equals the unit. If the ratio of fluor density to the fluor phase indicated as α, there would be three conditions as followed:

a) α=0 where the cell is empty of the fluor
b) α=1 where the cell is full of the fluor
c) 0<α<1 where the cell includes the common surface of both fluor phases.

According to the α level, the quantity of the variables in each control density were defined.

C. Biphase flow equilibrium in VOF model

1) The volume fraction equilibrium

Following the free level between two phases could be done using constant equilibrium solution for the volume fraction which would be as followed for q phase.

$$\frac{1}{\rho q}\left[\frac{\partial}{\partial t}\left(\alpha_q \rho_q\right) + \underset{\nabla}{\rightarrow}.\left(\alpha_q \rho_q \nabla_q\right) = \sum_{p=1}^{n}(\dot{m}_{pq} - \dot{m}_{qp})\right.$$
(8)

Where $\dot{m}_{pq}$ is the mass exchange from the p phase to the q one and $\dot{m}_{pq}$ is the mass exchange vice versa. When there are two fluors (e.g. the water free surface), the equilibrium could be solved just for one of the two fluros. With the second fluor, the fluor ratio could be concluded with the following limitation where n is the number of the fluor or phases.

$$\sum_{i=1}^{n} \alpha_i = 1 \ (9)$$

The transformation equilibrium properties of each cell would be resulted by the combination of the fluor properties of the cell. Take density biphase system as followed.

2) Momentum equilibrium

The momentum equilibrium could be solved for a flow in VOF model and the resulted velocity range is common for all the phases. The momentum equilibrium depends on the volume fraction and the physical properties (p, μ) of all phases:

ISSN No:-2456-2165

Numerical model specifications

$$\frac{\partial}{\partial t}\left(\rho \vec{\vartheta}\right) + \nabla.\left(\rho \vec{\vartheta}\vec{\vartheta}\right) = -\nabla p - \nabla.\left[\mu\left(\nabla\vec{\vartheta} + \nabla\vec{\vartheta}^{T}\right)\right] + \rho\vec{g} + \vec{F} \quad (10)$$

Where p is the pressure, v is the velocity vector, μ is the viscosity and ρ is the density. all the parameters might be defined based on the volume fraction of each phases. Also, F is the volumetric strain.

One of the relative common limitations is the difference of high velocities between phases. In this situation, the accuracy of the calculated velocities in the common surface may be affected.

There are various methods to solve this equilibrium but Geo reconstruction method was used in this study. The method is inconstant and the surface gradient between water and air is hypothesized for each flow cell linearly.

III. PHYSICAL MODEL CHARACTERISTICS

In order to check the accuracy of the results of this research, the physical model of the stepped spillway of the hydraulic laboratory of Savitribai Phule Pune University was used.. Similar approach has taken and the model made of plexiglass has 9 steps with a height of 1 cm and a length of 9 cm. The experiments were carried out in a flume with a length of 5 meters and a width of 9 centimeters. In measuring the discharge of a 90-degree over flow A depth gauge with an accuracy of 0.1 mm was used in the upstream and in taking the water level profile. Kamyab Moghaddam et al., conducted an experimental study on the stepped spillway, in their study, they used inclined steps with various slopes, equipped with liminimeter and pitot tube to measure the water depth, and record the velocities to calculate their relative energy losses by using the Bernoulli equation. Their results showed that this is a reliable method for the energy loss calculation. Taking inspiration from their approach, the recent study adopted a similar methodology to assess flow characteristics. Experimental results were then compared with those obtained by Kamyab Moghaddam et al., aiming to evaluate the accuracy of the model[8].

IV. NUMERICAL MODEL SPECIFICATIONS

Nowadays, the use of numerical and analytical methods in the study of the fluid environment has grown and developed, and due to the reliable results, they have been able to become a good substitute for physical models. Engineers, technologists, and scientists have employed numerical methods of analysis to solve a wide range of steady and transient problems. The fundamentals are essential in the basic operations of curve fitting, approximation, interpolation, numerical solutions of simultaneous linear and nonlinear equations, numerical differentiation and integration[9], [10]. The Flow 3D model is one of the most powerful models in the field of fluid dynamics, which has the ability to analyze the flow field in three dimensions. One of the capabilities of this program in the field of hydraulic analysis is the ability to use the fluid volume method in the field. It is a flow with a free surface It has solved the problems based on trial-and-error methods to a great extent.

In order to solve the flow field, this software uses the complete and three-dimensional equation of continuity and momentum, and with the help of the simultaneous solution of these equations, it calculates the characteristics of the flow field at each point. The software is capable of modeling five models of the model. It has the existing confusions Including the following:

Prandtl Mixing Turbulence Model: The simplest turbulence model is this software and it assumes that the viscosity of the fluid increases in areas with high shear stress (for example near the walls). In fact, this assumption for the flow Fully developed and stable In addition, this model assumes that the generation and decay of turbulence is true at all points of the flow.

One-equation turbulence transfer model: This model includes an equation for the transfer of turbulent kinetic energy, including terms of displacement and diffusion of turbulent kinetic energy.

The two-equation turbulence model: a complex and widely used model that includes two transfer equations for the kinetic energy of turbulence and its loss. This equation has coefficients with values of 1.92, 1.44 and 0.2.

Normalized node disturbance model or RNG: This model uses model equations, with the difference that in this model, coefficients are different from the model coefficients. In addition, the calculation method in this method is somewhat different from other methods and works very well in producing results in flows with low turbulence intensity and strong shear zones.

Turbulence model of large eddies: the main idea of this model is that all the turbulence structures that can be calculated using the computational grid are directly calculated and the small structures that cannot be calculated are approximated. In this model, the effects of turbulence, which are very small, are estimated with a turbulence viscosity that is proportional to the length scale multiplied by the speed fluctuations in that scale.

For meshing the model, square meshes with dimensions of 1 mm to 0.1 mm were used depending on the required accuracy. Also, the boundary conditions used in the model are given in Table 1.

TABLE I. BOUNDARY CONDITIONS USED IN THE MODEL

SPECIFIED PERESSURE	***The Upper boundary conditions***
OUT	Lower boundary condition
WALL	The lateral boundary condition of the flow
WALL	Boundary condition of channel floor
SYMMETRY	The boundary condition of the

ISSN No:-2456-2165

SPECIFIED PERESSURE	***The Upper boundary conditions***
	flow surface

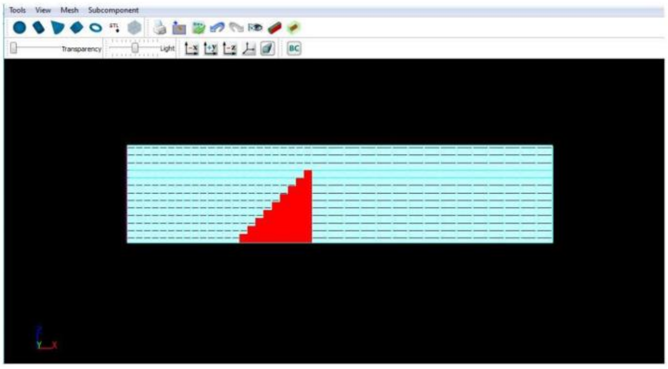

Fig. 3. A view of the model meshing method. *(figure caption)*

In order to determine the value of the accuracy and validity of the results, the values obtained for the water level profile from the numerical model were compared with their corresponding values in the comparative tests and their error values were obtained separately for the upstream and downstream of the Pelka structure. It is presented in tables 2 and 3.

TABLE II. COMPARISON OF THE ERROR VALUES OF THE UPSTREAM WATER LEVEL

Model name	***Software output***	***Laboratory output***	***The error percentage on the upstream***
Prentel mixing	0.107	0.104	2.8
An equation	0.107	0.104	2.8
	0.107	0.104	2.8
Normalized groups	0.106	0.104	1.89
Big eddies	0.106	0.104	1.89

TABLE III. COMPARISON OF THE ERROR VALUES OF THE DOWNSTREAM WATER LEVEL

Model name	***Software output***	***Laboratory output***	***The error percentage on the upstream***
Prentel mixing	0.107	0.104	2.8
An equation	0.107	0.104	2.8
	0.107	0.104	2.8
Normalized groups	0.106	0.104	1.89
Big eddies	0.106	0.104	1.89

V. RESULTS AND DISCUSSION

The results of numerical analysis of the model are given in figures 4 to 8. These figures show the amount of energy drop on the staircase structure at the water level above 6.10 cm. It is necessary to explain that the longitudinal axis of the flume length diagram, the transverse axis is the height of the structure. The specified points on the diagram are the values of the total energy of the water flow on the staircase structure.

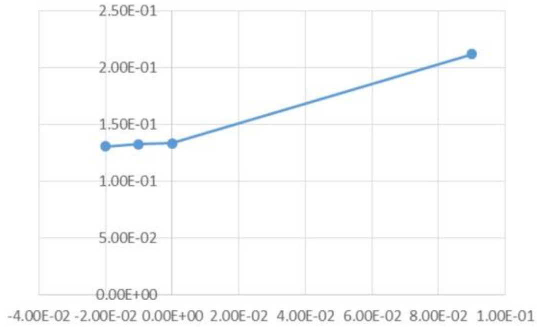

Fig. 4. Energy drop caused by Stepped Spillway 37%. Prandtl mixing Turbulence model

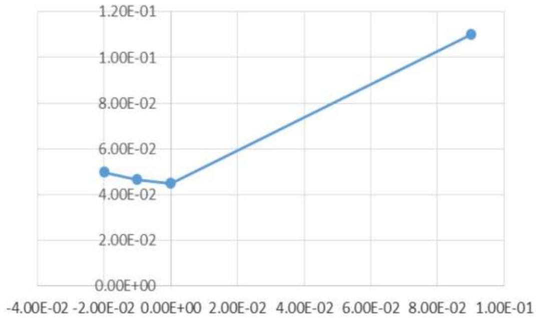

Fig. 5. Energy drop caused by Stepped Spillway 59%. One Equation Turbulence model

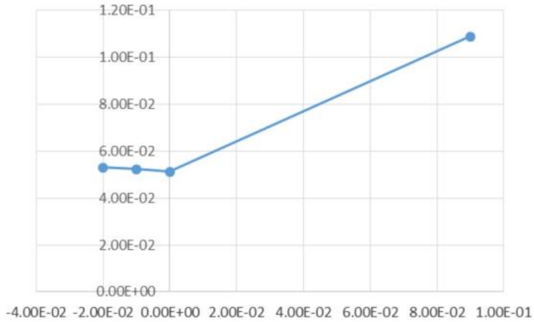

Fig. 6. Energy drop caused by Stepped Spillway 53%. k-ϵ Turbulence model

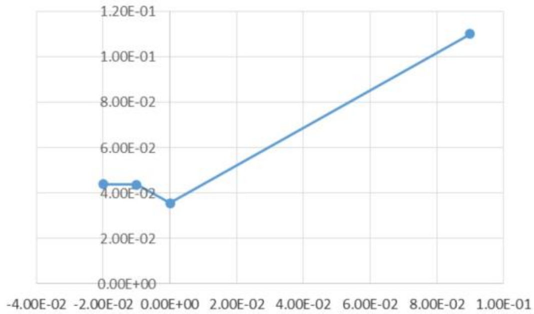

Fig. 7. Energy drop caused by Stepped Spillway 67%. Normalized Groups Turbulence model

ISSN No:-2456-2165

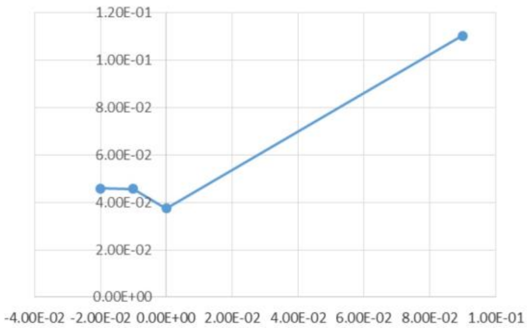

Fig. 8. Energy drop caused by Stepped Spillway 66%. Big Eddies Turbulence model

VI. GENERAL CONCLUSION

After simulating the flow, it was found that the model of normalized groups or RNG provides the closest values compared to the physical model. In the simulation of the flow upstream of the overflow due to low turbulence, no significant difference was observed between the turbulence models. In the simulation of the flow at the bottom of the spillway, a relatively large error was observed due to the high turbulence of the flow caused by the spill on the stairs. Prantel's mixing model and one equation have a high error due to solving the flow field with the help of one equation and are not recommended for flow conditions with high turbulence.

REFERENCES

[1]. C. Gualtieri and H. Chanson, “Physical and numerical modelling of air-water flows: An Introductory Overview,” *Environmental Modelling & Software*, vol. 143, p. 105109, Sep. 2021, doi: 10.1016/j.envsoft.2021.105109.

[2]. D. E. Reeve, A. A. Zuhaira, and H. Karunarathna, “Computational investigation of hydraulic performance variation with geometry in gabion stepped spillways,” *Water Science and Engineering*, vol. 12, no. 1, pp. 62–72, Mar. 2019, doi: 10.1016/j.wse.2019.04.002.

[3]. J. Chatila and M. Tabbara, “Computational modeling of flow over an ogee spillway,” *Computers & Structures*, vol. 82, no. 22, pp. 1805–1812, Sep. 2004, doi: 10.1016/j.compstruc.2004.04.007.

[4]. A. Bayon, J. P. Toro, F. A. Bombardelli, J. Matos, and P. A. López-Jiménez, “Influence of VOF technique, turbulence model and discretization scheme on the numerical simulation of the non-aerated, skimming flow in stepped spillways,” *Journal of Hydro-environment Research*, vol. 19, pp. 137–149, Mar. 2018, doi: 10.1016/j.jher.2017.10.002.

[5]. M. Tabbara, J. Chatila, and R. Awwad, “Computational simulation of flow over stepped spillways,” *Computers & Structures*, vol. 83, no. 27, pp. 2215–2224, Oct. 2005, doi: 10.1016/j.compstruc.2005.04.005.

[6]. M. Zhenwei, Z. Zhiyan, and Z. Tao, “Numerical Simulation of 3-D Flow Field of Spillway based on VOF Method,” *Procedia Engineering*, vol. 28, pp. 808–812, Jan. 2012, doi: 10.1016/j.proeng.2012.01.814.

[7]. N. D. Katopodes, “Chapter 12 - Volume of Fluid Method,” in *Free-Surface Flow*, N. D. Katopodes, Ed., Butterworth-Heinemann, 2019, pp. 766–802. doi: 10.1016/B978-0-12-815485-4.00018-8.

[8]. A. Kamyab Moghaddam, A. Hamedi, and S. Amirahmadian, “Experimental Study of Energy Loss in a Stepped Spillway Equipped with Inclined Steps in the Nappe and Skimming Flow Regimes,” *International Journal of Science and Engineering Applications*, vol. 11, no. 12, pp. 346–350, 2022, doi: 10.7753/IJSEA1112.1031.

[9]. A. K. Coker, “CHAPTER 1 - Numerical Computation,” in *Fortran Programs for Chemical Process Design, Analysis, and Simulation*, A. K. Coker, Ed., Houston: Gulf Professional Publishing, 1995, pp. 1–102. doi: 10.1016/B978-088415280-4/50002-9.

[10]. M. Dashtibadfarid and M. Afrasiabi, “Low-Permeability Concrete: Water-to-Cement Ratio Optimization for Designing Drinking Water Reservoirs,” *International Journal of Innovations in Engineering and Science*, vol. 2, no. 11, 2017.