ISSN 2229-5518

Numerical Investigation of Energy Dissipation in Ogee Spillway

Smitha Mohan K, Greeshma Elandassery

Abstract— Spillway is one of the most significant components of a dam to ensure the safety during the flood. The identification of hydraulic characteristics for a spillway is difficult issue because of the rapidly varied flow type. Computational fluid dynamics (CFD) is a powerful tool for investigating system relating fluid flow problems by means of computer based simulation techniques. ANSYS FLUENT is one of the packages used widely for solving water resource related fluid problems. This paper aims to undergo a two dimensional model study of flow simulation in ogee spillway and energy dissipation using Fluent 16.0. Flip bucket is used as mode of energy dissipation in this study.

Index Terms—Ansys- Fluent, CFD, Energy dissipation, Flip-bucket, numerical modeling, Ogee Spillway, and physical model.

1 Introduction

Spillways and other flood outlets are designed to safely convey floods to the watercourse downstream from the dam and to prevent overtopping of the dam. Spillway structures are so complicated to study and analyze the hydraulic behavior. Ogee spillway are the most important structures used to control flood water and built at the same time concrete or masonry dams are constructed. Spillways are hydraulically sized to safely pass floods equal to or less than the current critical Probable Maximum Flood (PMF), [U.S Design Standards, 2010]. The development of computer science and different types of computational fluid dynamics (CFD) software, the behavior of ogee spillways can be studied in a short time and without paying high expenses. In this study, FLUENT software has been used to simulate flow over ogee spillway to study pressure variations and results were compared with experimental results.

Different studies based on CFD modeling are going on in this field. CFD modeling is used to Simulate Spillway's approach channel is done at the guide wall of the Kamal Saleh Dammodel by flow3d in which they explained that the geometry of guide wall (left) caused the instability in the flow pattern and secondary vortex flow at the beginning of approach channel. They found shape reduced the performance of the weir to remove the peak flood discharge and the behavior of water in spillway is strongly affected by the flow pattern formation [5].

To Eray Usta [1] did numerical investigation of hydraulic characteristics of Laleli Dam spillway and compared with physical model study. Flow 3d is used as a CFD tool to obtain the hydraulic parameters of Lalalei Dam spillway which is programmed to use RANS equations. Grid independency is found to be important. Flow over spillway with high mesh resolution near wall is found to be significant to advert flow accurately and pressure distribution. The impact of meshing in numerical studies using software are evaluated by comparing Tri pave and Quad pave meshing and concluded quad pave meshing as better [3]. The appropriateness of revised design better than the existing was explained using Flow3d model to investigate Improvement of hydraulic stability for spillway using CFD model [4]. Different turbulence model are analyzed to predict the hydraulic condition of flow over the Ogee spillway which showed an increased efficiency is obtained for RNG- K Epsilon turbulence model using fluent software [2]. From the literature review it is found remarkable to conduct a numerical study on ogee spillway to analyze the hydraulic behavior in less time. Moreover the adaptability of Computational Fluid Dynamics in this field can be notified for further studies. Computational fluid dynamics is a tool for investigating system relating fluid flow problems by means of computer based simulation techniques. Simulation of flow is based on solution of Navier – Stokes and continuity equations. These equations are discretized and solved at each computation cell. The solution to fluid dynamics problem typically involves calculating various properties such as velocity, pressure, density, temperature as function of space and time based on following principles (ANSYS, 2009). Equation (1) represents the continuity or mass of conservation, Where S_m is mass, ρ is fluid density and v is fluid velocity.

$$\frac{\partial p}{\partial t} + \Delta(\rho v) = S_m \qquad (1)$$

Conservation of Momentum: the statement for this law (F=ma) is given by Navier Stokes equation (2).

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_i} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2\partial u_k}{3\partial x_k}\delta_{ij}\right)\right] + \rho g_i + F_i \qquad (2)$$

First term on left side represents the variation of fluid momentum in time; second term represents transport of momentum in the flow. First term on right side represents effect of gradients in the pressure P, second term; transport of momentum due to molecular viscosity, third term; effect of gravity and the last term represents sum of all forces acting on fluid.

- *Smitha Mohan K is currently working as assistant professor in Civil engineering at Govt. Engineering College, Thrissur, KTU University, India, PH-919526946763. E-mail: smitha.mohanam@gmail.com*
- *Greeshma Elandassery is currently pursuing masters degree program in water resources & hydroinformatics inGovt. Engineering College, Thrissur, KTU University, India, PH-918547290659. E-mail: greeshmaranjith1@gmail.com.*

ISSN 2229-5518

Before solving the fluid flow problems, Fluent needs the domain in which the fluid flow take place which has to be discretized into smaller control volumes. The software uses design modular session for creating geometry for the fluid flow domain. These geometry of domain then need to be meshed to form a grid of triangles or quadrilaterals which could either be structured or unstructured. After completing pre-processing stage, fluent setup and solver stage is required before simulation. In setup stage details of material property, boundary conditions etc. are provided and in solver part solution methodology, calculation activity and flow time specification can be set. After fluent simulation results can be verified in the post processing segment.

2 NUMERICAL STUDY

2.1 Validation work

In the present study, experimental findings of Laleli dam ogee spillway are used in order to calibrate the numerical models. A scheme of the ogee spillway is depicted in Fig.1. The geometric parameters of the investigated ogee spillway are as follows, (Eray Usta thesis report METU University, 2014): Maximum water level of dam : 1480m, Crest level of spillway : 1473.5m, Spillway Crest length : 38m, 4 radial gates, Elevation difference (spillway crest & flip bucket deflector) : 103.5m, Slope of spillway channel : 1.43.

Fig.1.Laleli Dam Ogee Spillway

Using the available model details geometry of ogee spillway is prepared as shown in Fig.2.

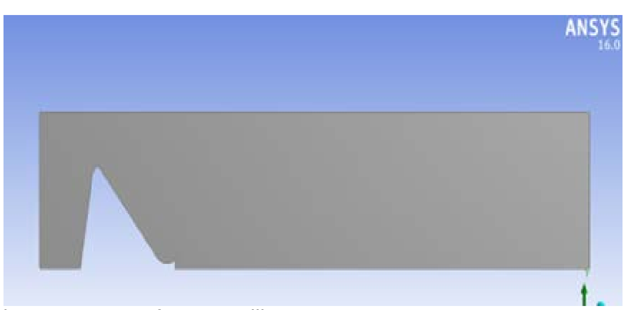

Fig.2 Geometry of Ogee Spillway

Grid generation is the relevant part of Ansys Fluent because fine structured mesh enables more accurate solution to be generated. Meshing is finalized with the standard deviation values of element quality (4.9708e-002), Aspect ratio (4.7763e-002) and orthogonal quality (1.2888e-002). Fig.3 shows the meshed ogee spillway model. Total nodes are 17124 and elements are 16768.

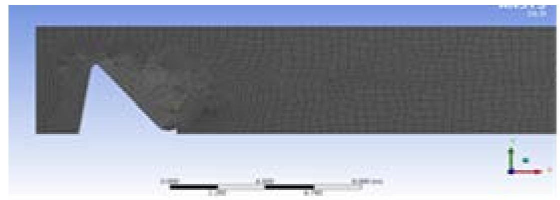

Fig.3. Meshed Ogee Spillway

Analysis part starts with general tab in which pressure based solver and transient state is selected. Multiphase model with VOF model type and number of phases as 2 (air and water) is marked. Implicit method is chosen to increase solution accuracy.

In general, one of the most important phases of the numerical analysis of flow field is to determine proper boundary conditions, which are matched appropriately with the physical conditions of the problem. In this study for numerical modeling of ogee spillway, the named parts have been provided as in Fig.4. According to Fig.4, the original boundary conditions of flow over ogee spillway include inlet (velocity inlet for water, and pressure inlet for air), outlet (pressure outlet), and stationary wall (wall). All tables and figures will be processed as images. You need to embed the images in the paper itself. Please don't send the images as separate files.

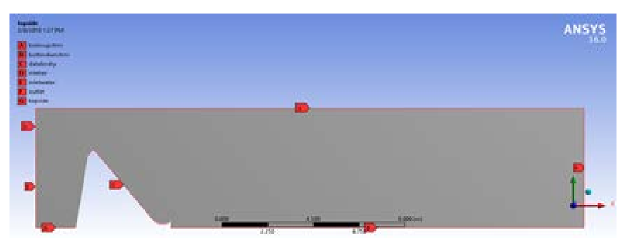

Fig.4. Named parts of fluid domain

The solution part is initialized and patching is done to set initial water level. Fig.5 provides the patched image.

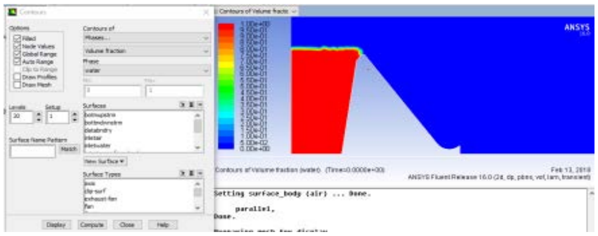

Fig.5 water level patched before simulation

Water volume fraction after required time step is noted as in Fig 6.

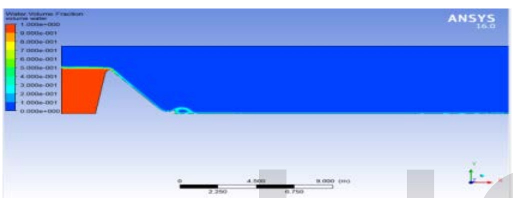

Fig.6water level patched before simulation

2.2 Study relating flip bucket exit angle and Energy dissipation

Energy dissipator is a device constructed in a waterway to reduce the kinetic energy of fast flowing water. Different types of energy dissipators may be used along with a spillway, alone or in combination of more than one, depending upon the energy to be dissipated and erosion control required downstream of a dam. Broadly, the energy dissipators are classified under two categories – Stilling basins or Bucket Type. The flip bucket energy dissipater is suitable for sites where the tail water depth is low (which would require a large amount of excavation if a hydraulic jump dissipater were used) and the rock in the downstream area is good and resistant to erosion. In the present study an attempt is made to investigate relation between exit angles of flip bucket to energy loss.

Model details are used from validation study with an existing exit angle of 43.5º and discharge of 181l/s. Study is done for various discharges 180l/s, 200l/s, 250l/s, 300l/s and 350l/s at angles of 30º, 40º, 45º, 50º.

3 Results

Measurement points were spotted along the ogee spillway as in Eray Usta thesis report (2014) to make comparison study. Fig.7 shows location of these measurement points on which pressure head is noted. From the comparison validation is achieved for adopting Ansys Fluent as an effective CFD tool to solve fluid flow problems. Pressure variation on the model spillway was examined to be more accurate and relatable to the experimental results. A relative study on exit angle and energy dissipation of Flip bucket is done using the same model by changing the model conditions.

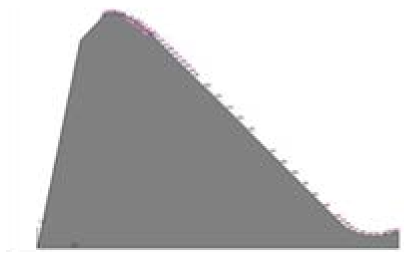

Fig.7.Measurement points along spillway

Pressure head values in terms of cm of water column are measured at measurement points and the same is compared with the experiment and numerical (Flow-3D) results of Eray Usta (2014) thesis report as shown in Fig.8.

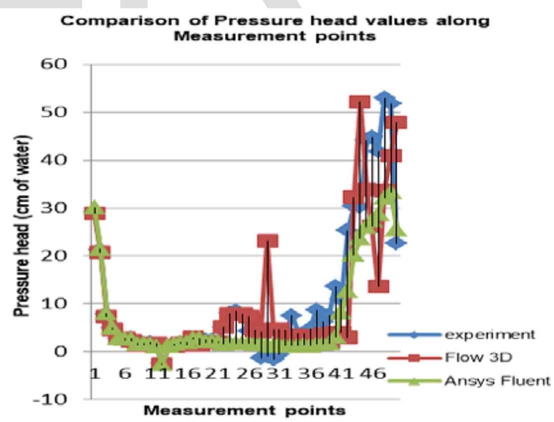

Fig.8.Comparison of pressure head values

From Fig.8 it can be noted that numerical investigation of pressure head values along ogee spillway using Fluent is parallel to the results from reference. Slight variations are due to the difference in meshing and solver methods.

ISSN 2229-5518

Studies relating flip bucket exit angle and energy dissipation is done to find out a best angle of flip bucket for the prepared model that would give maximum energy dissipation in less projectile path. From the numerical simulation for each case, angle of range 40º to 45º can be selected as optimum case. At angle of 45º maximum energy dissipation is obtained for a flow of 200l/s. And minimum projectile length with maximum energy dissipation is obtained at angle of 45º. The observed values of each simulation are tabulated (see Table 1).

TABLE 1
TABULATED OBSERVED VALUES

		Exit angle			
		30º	40º	45º	50º
Discharge (l/s)	180	5.732	178.94	257.8	214.52
	200	18.19	158.46	268.05	197.78
	250	10.02	117.03	232.361	142.11
	300	4.802	90.06	157.23	131.20
	350	8.542	85.85	148.031	115.14

To make studies more inferable plots are made and simulation results are verified. Fig.9 represents the relation between exit angle of flip bucket to energy dissipation and length of projectile flow from the bucket tip to tail water respectively for varying discharge.

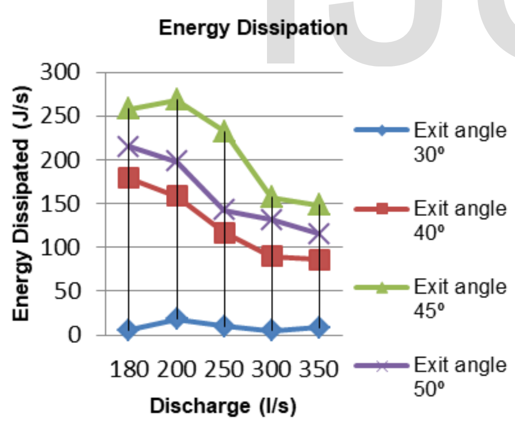

Fig.9 Energy dissipation plot in the study

4 CONCLUSION

In the present numerical study, flow over ogee spillway was simulated by using 2D model in FLUENT software. The VOF model was used with two phase for simulation. Results obtained in this study showed that the flow parameter such as the pressure head obtained by the FLUENT numerical model for were in good agreement with the experimental results and Flow3D results for flow over ogee spillway and can be used for analysis or design of such hydraulic structures. Study relating exit angle of flip bucket flow for various discharge was done and an optimum angle of 45º for the exit angle is obtained. Accuracy of results mainly depends on fineness of meshing and has affected this study results. Numerical models are usually much less time consuming than physical models. Furthermore, the computational cost of numerical models is low relative to the experimental tests. In a numerical model, changes in a design can be easily adopted to existing model. High-capacity computers and efficient CFD codes provide realistic fluid flow solutions so that CFD could be thought as virtual laboratory. Ansys could generate much efficient solution for fluid flow problems so that computation becomes more relatable to experimental work in less time

ACKNOWLEDGMENT

The authors heartully wish to thank institution head, Dr. N. Sajikumar for motivation and guidance and Mr. Roopesh Kaimal for his support and help to conduct this study.

REFERENCES

[1] Aydın, I., Göğüş, M., Altan-Sakarya, B.A., Köken, M., (2012), *Laleli Barajı Ve Hidroelektrik Santrali Dolusavak Hidrolik Model Çalışmaları, İnşaat Mühendisliği Bölümü, Hidromekanik Laboratuarı,* ODTÜ. Eray Usta Thesis report based on this journal.

[2] DR. Shaymaa Abdul Muttaleb Alhashimi, (2013), *CFD modeling of flow over ogee spillway by using different turbulence models*, International Journal of Scientific Engineering and Technology Research, Vol 2, Pp. 1682-1687.

[3] Hamiddreza..Vosoughifar,..Alireza..Daneshkhah (2011), *Numerical investigation of passed flow different parameters over a standard ogee spillway to satisfy flow profile in cfd method* , Conference paper (5thSASTech), Khavaran Higher-education Institute, Mashhad, Iran.

[4] Kim, S., An, S., (2010), *Improvement of hydraulic stability for spillway using cfd model*, International Journal of the Physical Sciences Vol. 5(6), pp. 774-780.

[5] Sadegh Dehdar Behbahani, Abbas Persaie, (2016), *Numerical modelling of flow pattern in dam spillway's guide wall – case study* balaround dam, iran, Journal of Alexandria Engineering, 55; 467-473.