Journal of Applied Mechanical Engineering
ISSN: 2168-9873

Date et al., J Appl Mech Eng 2017, 6:5
DOI: 10.4172/2168-9873.1000287

Research Article
Open Access

Numerical Modeling of Flow over an Ogee Crested Spillway under Radial Gate: VOF and MMF Model

Date V[1], **Dey T**[1]* **and Joshi S**[2]

[1]*Department of Mechanical Engineering, D.Y. Patil College of Engineering, Akurdi, Pune 411044; Savitribai Phule Pune University, Pune, India*
[2]*Water Resource Department, Dapodi, Pune, India*

Abstract

The present research work shows the simulation of the gated flow over an ogee crested spillway for one of water reservoir. The average velocities and Froude Number analysis at various gate openings gives a better insight of flow behavior. Also, the simulations were carried out by changing the gate bottom shape. The STAR CCM+ CFD tool is used to solve the fluid flow performance. The flow parameters near the bottom of the gate have been studied with two types of fluid flow models i.e. Volume of Fluid (VOF) and Multi-Mixture Fluid Models. The use of Volume of Fluid (VOF) multiphase model together with RNG k-ε turbulence for the simulation, gives the excellent agreement between the experimental and numerical data. The spillway performance of the gated flow at various gate openings resembles with the actual flow behavior. The applicability of the CFD model to simulate the gated flow over ogee crested spillway is reviewed. The computational model study showed that CFD can be useful in hydraulics structures for designing of various reservoirs. This numerical model gives significant advantage in practice, in terms of parametric studies.

Keywords: Ogee; Star CCM+; Volume of fluid (VOF); Spillway; Multi-mixture fluid (MMF)

Introduction

Due to recent advances in computing technologies, numerical modeling of hydraulic structures is becoming increasingly important in the engineering field, to the point where these models frequently replace the former industry standard of scaled physical modeling. This replacement is due to certain advantages that are associated with numerical modeling. Numerical models are often much less expensive than physical models because they require no laboratory space, no materials or construction and can be easily modified to accommodate design changes. All that is required for simulations is the computer, the software and the engineering know-how to interpret the results. Although many numerical models exist, validation data is often difficult to obtain and therefore, there is always a level of uncertainty associated with results.

Information regarding the flow of water over spillways has historically been obtained through the use of physical model experiments. Hydraulics experts are interested in CFD and are eager to verify the capability of the numerical modelling software.

Femando [1] gives only real-time predictions of the discharge in any situation of energy head and gate opening within the operation range of reservoir. Riyadh [2] uses CFD tool for validating the experimental analysis with numerical model for flow velocity and pressure. The study of flow rate, water surfaces and crest pressure was carried out by Kim [3] with FLOW-3D CFD tool and concludes the acceptable numerical errors. Chanel and Doering [4] represent the comparison between gated and free opening flows with FLOW-3D and concluded that CFD should not be considered a complete replacement for physical modelling. Dan Gessler [5] overcomes this issue and proposes that FLOW-3D can be used to simulate the flow over spillway. Jean Chatila [6] used the k-ε turbulent model and predicted the reasonable results that are consistent with general flow characteristics over spillway. Fatema [7] simulate flow over an ogee spillway by a commercial numerical model and investigate the ability of the model predict several characteristics of flow. The numerical modelling shows efficiency in studies due to saving in time and money and ability of monitoring all necessary data in several conditions. Bruce M. Savage [8] studied the physical model, numerical model and existing literature. Discharge and pressure data were recorded for 10 different flow conditions which gives reasonably good agreement between physical and numerical models for both pressures and discharges (Figure 1).

Robert [9] did the study on hydraulic model for evaluating flow conditions contributing to abrasion damage in the stilling basin. This study recommended that the spillway gates be operated uniformly for reducing abrasion. James Higgs [10] studied vortices using CFD model and stated that identical flows through each bay will reduce the strength of vortices. Bhajantri [11] evaluated flow over a spillway using two-dimensional finite volume based numerical model which gives satisfactory results between numerical modeling and physical modeling. Also Ho [12] used the CFD technique for modeling spillway which shows good agreement between published data. Here further investigation will be carried on influence of turbulence flow, non-

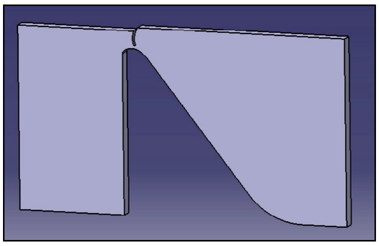

Figure 1: CAD model of fluid domain.

***Corresponding author:** Dey T, Department of Mechanical Engineering, D.Y. Patil College of Engineering, Akurdi, Pune 411044; Savitribai Phule Pune University, Pune 411007, India, Tel: +91 20 27653058; E-mail: tapobrata.dey@gmail.com

Received August 11, 2017; **Accepted** October 03, 2017; **Published** October 07, 2017

Citation: Date V, Dey T, Joshi S (2017) sNumerical Modeling of Flow over an Ogee Crested Spillway under Radial Gate: VOF and MMF Model. J Appl Mech Eng 6: 287. doi: 10.4172/2168-9873.1000287

uniform upstream flow and adjacent pier structures. The study carried out by Sadegh [13] showed that optimal design of the guide wall leads to increase the performance of the spillway to pass the flow smoothly. Sebastian [14] studied pressure and velocity at the crest of the spillway and validated the data with experimental data (Figure 2).

Numerical Modeling

Model setup for computational domain can be separated into three main subcomponents as geometry definition, grid definition and boundary conditions. Also it is necessary to define the overall model physics related to the fluid properties. In the numerical modeling three steps that are necessary to define: geometry definition, grid generation and boundary condition. The geometry definition of the model has many similarities with physical model. The software used for creation of geometry is Computer Aided Three-dimensional Interactive Application (CATIA) V5 R15 due to its simplicity and convenience in part designing, surfacing, wireframe designing, assembly, etc.

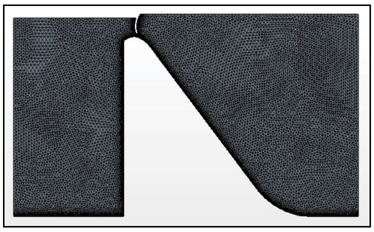

Figure 2: Surface meshing of fluid domain.

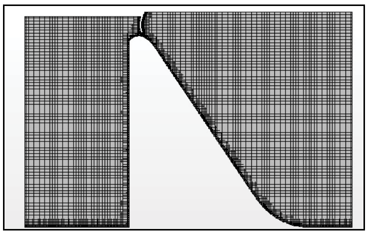

Figure 3: Volume mesh of fluid domain.

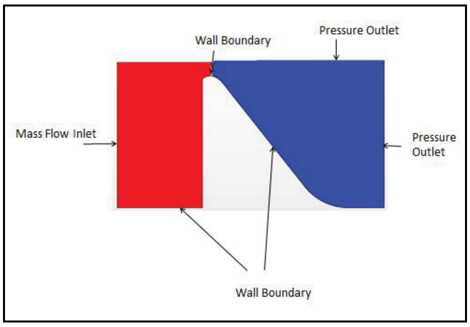

Figure 4: Boundary conditions for simulation model.

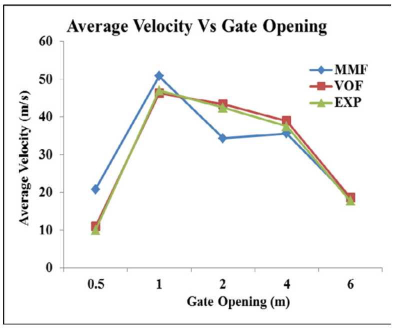

Figure 5: Average velocity variation at different gate openings.

Definition of the grid is a significant aspect of the model development. An area with rapidly varying geometry, such as the wing of walls of the spillway, requires small computational cells for accurate resolution. Surface triangles are created on the "CFD ready CAD" surfaces for all configurations with the help of inbuilt meshing of STAR CCM+ software package (Figure 3).

Flow variations are important near gate surface and near the wall of dam. Thus, in order to capture the flow physics at such locations more precisely, mesh refinement is done. The total number of shell elements or surface triangles is approximately 117544.

A 3-D volume mesh was generated using STAR CCM+ after importing the fluid volume in the solver. A 3-D volume mesh consisting of block-structured meshing like cubical blocks placed side-by-side. In the STAR CCM+ CFD software package the block-structured grid is named as Trimmer and we are using this mesh for the volume meshing in the simulation.

The boundary conditions for solving the problem were defined as shown in Figure 4. The left side of the domain was set as mass flow inlet in this software. Ogee spillway construction and the lower part of the domain were set as the wall boundary conditions. Both the air boundaries at right hand side of the domain were set to pressure outlets. The pressure outlets were initially assumed to be at an atmospheric. The boundaries near the radial gate i.e. gate boundaries were set to the wall boundary as the gate was fixed at some opening. Wall boundary conditions in this problem were all set to the stationary, no-slip wall.

Fluid Flow Modeling

Volume of fluid (VOF) model

The volume of fluid model is based on the fact that two phases of flow problem i.e. air and water was not to be interpenetrating them. Here the sum of volume fractions in each cell was unity. As the different volume cells of the domain were shared by either the phases or single phase. Therefore different variables and properties of each volume cell is the function of volume fraction of one of the phases or combination of both the phases, depending on the values of volume fraction. The different values of properties and variables at each cell were calculated from the advection of the fluid at each cell face.

For solving the fluid domain, a RNG k-ε momentum equation was solved throughout the domain. In which the value of different variables and properties were calculated at each cell face. The resulting velocity field was distributed among all the phases in the domain. The momentum equation given below is depends on the volume fractions of all phases through the properties ρ and μ.

$$\frac{\partial}{\partial t}(\partial \vec{u})+\nabla(\partial \vec{u}\vec{u})=-\nabla p+\left[\mu\left(\nabla \vec{u}^{T}\right)\right]\rho\vec{g}+\vec{F} \tag{1}$$

Multi-mixture flow (MMF) model

The MMF model is suggested to use for bubbly flows and pneumatic transport. It 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 coupling between the phases should be strong. It can also be used to model homogeneous multiphase flows with very strong coupling and the phases moving at the same velocity. The mixture model can model n phases by the continuity equation for the mixture, the momentum equation for the mixture, and the volume fraction equation for the secondary phases, as well as algebraic expressions for the relative velocities.

Continuity equation:

$$\frac{\partial}{\partial t}\left(\rho_m\right)+\nabla\left(\rho_m\vec{u}_m\right)=0 \tag{2}$$

Momentum equation:

$$\frac{\partial}{\partial t}\left(\rho_m\vec{u}_m\right)+\nabla\left(\rho\vec{u}_m\vec{u}_m\right)= -\nabla p+\nabla\left[\mu_m\left(\nabla\vec{u}_m+\vec{u}_m^{T}\right)\right]+\left[\sum_{k=1}^{n}\alpha_k\rho_k\vec{u}_{dr,k}\vec{u}_{dr,k}\right] \tag{3}$$

It is a relatively recent development from the standard k-model is RNG k-model. The RNG turbulence model solves for turbulent kinetic energy (k) and turbulent kinetic energy dissipation rate (ε). The RNG-based models rely less on empirical constants while setting a framework for the derivation of a range of parameters to be used at different turbulence scales.

Results and Analysis

Validation of the average velocity variation

The experimental data of the velocities at various gate openings is validated against the VOF model and MMF model in which VOF model gives the close agreement as compared to the MMF model. Figure 5 showed a comparison of the experimental data and the numerical model as described in the Table 1.

Gate Opening (m)	Average Velocity (m/s)			Difference (%)	
	Experimental	VOF	MMF	VOF	MMF
0.5	10	10.955	20.799	8.7	107.99
1	47	46.28878	50.789	1.5	8.06
2	42.5	43.36421	34.307	2	19.27
4	37.5	38.93685	35.628	3.7	4.99
6	17.7	18.5856	18.478	4.8	4.39

Table 1: Average velocity comparison near gate bottom of gated spillway flow.

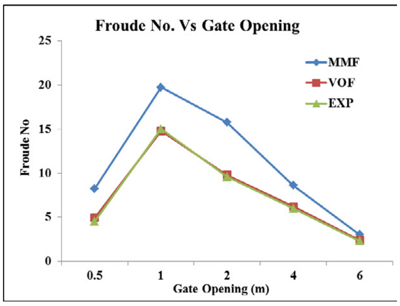

Figure 6: Froude number variation at different gate openings.

Gate Opening (m)	Froude Number (F_r)			Difference (%)	
	Experimental	VOF	MMF	VOF	MMF
0.5	4.52	4.946	5.2510	9.55	16.29
1	15	14.779	16.743	1.51	11.57
2	9.6	9.79	11.759	2.03	22.55
4	6	6.22	7.5969	3.83	26.90
6	2.31	2.423	3.0233	5.0	31.04

Table 2: Froude Number (Fr) comparison near gate bottom of gated spillway flow.

As seen in Figure 5 there is excellent agreement between the predicted and measured average velocities for the VOF model which were never exceeding more than 9%. The difference observed between these predicted and numerical model were due to the approximations made in the simulation model and also due to small errors in the water level at gate openings, caused by the spill region at inlet boundary. The MMF model shows the large percentage of errors so we preferred VOF model for the simulations (Figure 6).

Validation of the froude number (F_r) variation

The Froude number (Fr) variation near the gate bottom is found to be supercritical for all gate openings. The Froude number (Fr) analysis was carried out by both the fluid flow modeling's i.e. by VOF model and MMF model. The results obtained showed that the difference between the experimental data and the VOF model shows better agreement than the MMF model. From Table 2 it is shown that difference between VOF and experimental lies below 9% and that of between MMF and experimental lies above the 9% for this spillway problem. Hence the VOF model should be suitable for the fluid flow modeling rather than MMF Model.

Also Froude number (Fr) for the higher gate openings are lesser than the small gate openings. It shows that the flow is more supercritical at small gate openings than that of higher gate openings. The flow tends to critical region as we go on increasing the gate openings. Hence gates should be operated at higher gate openings.

Conclusion

A numerical model using VOF multiphase flow model together with RNG k-ε turbulence model is more suitable than MMF model to simulate the flow over an ogee crested spillway with gated flow. The data obtained from large scale experiments of dam reservoir verifies the VOF model data more significantly.

The Froude Number variation after 1m gate opening approaches towards unity i.e. flow tends to be critical. So we must not keep the open at 1m opening for long time as the flow is more supercritical in this region it should be operated at 6`m or full gate opening for dam reservoir.

The above study showed that CFD can be viewed as better design tool for hydraulic structures with proper analysis for validation. Number of cases could be easily simulated which provide us the information about the various flow parameters such as velocity, flow, pressure, and another parameter associated with dam flow. Finally, the numerical model has many advantages in practice, in terms of parametric study.

References

1. Salazar F, Morán R, Rossi R, Oñate E (2013) Analysis of the discharge capacity of radial-gated spillways using CFD and ANN –oliana dam case study. J Hydraulic Res 1: 1-9.
2. Al-Zubaidy RZ, Muttaleb AS, Alhashimi H (2013) Numerical simulation of two-phase flow over mandali dam ogee spillway. Int J Structural and Civil Eng Res 2: 1-13.
3. Kim DG, Park JH (2005) Analysis of flow structure over ogee-spillway in consideration of scale and roughness effects by using CFD model. KSCE J Civil Eng 9: 161.
4. Chanel PG, Doering JC (2008) Assessment of spillway modeling using computational fluid dynamics. Canadian J Civil Eng 35: 1481-1485.

5. Gessler D (2005) CFD modeling of spillway performance. J Impact of global climate change pp. 1-10.

6. Chatila J, Tabbara M (2004) Computational modeling of flow over an ogee spillway. J Computers and Structures 82: 1805-1812.

7. Goharrizi FZ, Moghadam MA (2010) Evaluation of a numerical modeling for flow over an ogee spillway. 8th International River Engineering Conference Shahid Chamran University pp. 110-116.

8. Savage BM, Johnson MC (2001) Flow over ogee spillway: Physical and numerical model study. J of Hydraulic Eng 127: 640-649.

9. Einhellig RF (1999) Hydraulic model study of folsom dam spillway performance and stilling basin abrasion. US Department of the Interior Bureau of Reclamation pp. 125-131.

10. Higgs JA (1997) Folsom dam spillway vortices computational fluid dynamic model study. US Department of the Interior Bureau of Reclamation pp. 31-37.

11. Bhajantri MR, Eldhoand TI, Deolalikar PB (2006) Hydrodynamic modelling of flow over a spillway using a two-dimensional finite volume-based numerical model. Sadhana 31: 743-754.

12. Ho DKH, Boyes KM, Donohoo SM (2001) Investigation of spillway behavior under increased maximum flood by computational fluid dynamics technique. 14th Australasian Fluid Mechanics Conference Adelaide University pp.10-14.

13. Dehdar-behbahani S, Parsaie A (2016) Numerical modeling of flow pattern in dam spillway's guide wall. Case study: Balaroud dam, Iran. Alexandria Eng J 55: 467-473.

14. Erpicum S, Peltier Y, Dewals B, Archambeau P, Pirotton M (2017) Pressure and velocity on an ogee spillway crest operating at high head ratio: Experimental measurements and validation. J Hydro-environment Res pp. 1-9.