# International Engineering Research Journal
# Numerical Modeling of Flow over Ogee Crested Spillway under Radial Gate

**Vishal Date[1], Tapobrata Dey[1] and Shashikant Joshi[2]**

[1]Mechanical Engineering Dept., D Y Patil College of Engineering Akurdi, SPPU, Pune, India
[2]Water Resource Department, Govt. of Maharashtra, Dapodi, Pune, India

***Abstract***

*This research shows the simulation of the gated flow over an ogee crested spillway for one of waterreservoir. The average velocities and Froude Number analysis at various gate openings gives a better insight of flow behavior. The STAR CCM+ CFD tool is usedto solvethe fluid flow performance. 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 performanceof 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.*

## 1. Introduction

Ogee-crestedspillway is themost popular hydraulic structures in studies, because of its fine hydraulic characteristics. Engineers widely use it in situations and know if properly designed. It has ability to pass flows efficiently and safely, with relatively good flow measuring capabilities. Researcher need to understand flow characteristics changing due to variety of hydraulic and geometry design modifications such as hydraulics head, make them to use modelling, both physical and numerical. These days computational fluid dynamics (CFD) is used extensively for developing model and analyse complex issues related to hydraulic design, planning studies for future generation stations, civil maintenance, supply efficiency, and dam safety. However it is necessary to evaluate computational models with experimental models for solving the governing equations of fluid flow.

In the recent years several attempts provided for CFD solutions and mathematical methods for flow optimization. In some of them there is a good agreement with experimental data for a limited numbers of their solutions. One of them did it by potential flow theory and mapping into the complex potential plane. This attempt led to conclude that viscosity has a negligible influence on the location of the free surface. Also another one obtained that the minimum pressure point for a given head is up to the boundary condition configuration. And a good one showed excellent agreement for water surfaces and discharge coefficients for a limited number of flows. However, pressure data are only recorded at five locations downstream from a nonstandard crest at one flow and showed some variability.

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 (2013) gives only real-time predictions of the discharge in any situation of energy head and gate opening within the operation range of reservoir. Riyadh (2013) 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 (2005) with FLOW-3D CFD tool and concludes the acceptable numerical errors. Chanel and Doering (2008) 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 (2005) overcomes this issue and proposes that FLOW-3D can be used to simulate the flow over spillway. Jean Chatila (2004) used the k-ε turbulent model and predicted the reasonable results that are consistent with general flow characteristics over spillway. Fatema (2010) 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 (2001) studied the flow over an ogee spillway by constructing 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.

Robert (1999) 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 (1997) studied vortices using CFD model and stated that identical flows through each bay will reduce the strength of vortices. M. R. Bhajantri(2006) evaluated flow over a spillway using two-dimensional finite volume based numerical model which gives satisfactory results between numerical modelling and physical modelling. Also D K H Ho(2001) used the CFD technique for modelling spillway which shows good agreement between published data. Here further investigation will be carried on influence of turbulence flow, non-uniform upstream flow and adjacent pier structures.The study carried out by Sadegh (2016) showed that optimal design of the guide wall leads to increase the performance of the spillway to pass the flow smoothly. Sebastian (2017) studied pressure and velocity at the crest of the spillway and validated the data with experimental data.

The different studies carried out on spillway all gives the better agreement between numerical modelling and physical modelling. This work explains about the flow analysis at different gate openings and keeping the water head constant. Also studied the behaviour of average velocities from the discharges and Froude Number variation at different gate openings near the gate bottom.

## 2. Objectives

Analysis of the flow over a radial gated ogee crested spillway for average velocity distribution and Froude no variation.

Validation of the results obtained with the experimental data.

## 3. Model Description of CFD

For the study of numerical performance of radial gated flow over the ogee crested spillway STAR CCM+ software package was used. For solving the radial gated flow over an ogee crested spillway this software used the VOF model (Volume of Fluid model). This VOF model was introduced by Nichols and Hirt (2001) for solving many hydraulic problems and for modeling the two-phase flow models in many hydraulic structures. In this problem most of the volume of cell contains water and remaining one air. Before applying the VOF model in STAR CCM+, geometry was developed in the auxiliary software CATIA V5 and thereafter geometry was imported in the STAR CCM+ for simulation. For solving this problem in the software, the equations of governing flow which includes continuity equation (law of mass conservation) and Navier-Stokes equation (momentum conservation) were used. Solution of the Navier-Stokes equation was obtained by using k-ε turbulence from the Reynolds-Averaged Turbulence model. Thus the governing equations were determined.

### 2.1 Boundary Conditions

The boundary conditions for solving the problem were defined as shown in Fig. 1. 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 atan 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. As the water as fluid flowing nears the wall the viscosity near the wall dealt with it. Here CFD tool dealt with multiphase flow with two materials as air and water so the segregated solver was used. All this was modeled under the VOF model for multiphase flow.

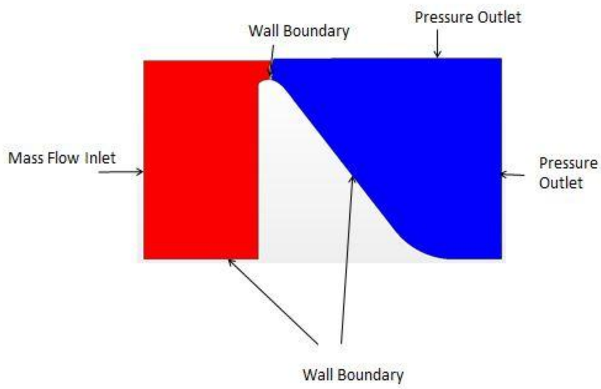


**Fig. 1**Boundary conditions for simulation model

## 4. Fluid Flow Modeling

### 4.1 Volume of Fluid Model (VOF)

The volume of fluid model is based on the fact that two phases of flow problem i.e. air and water were 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. Firstly in this case model had to calculate the position of partially-filled cell by relative linear interface. Secondly by using the normal and tangential distribution of velocity were calculated of the advection of the fluid. And lastly the volume fraction in each cell was calculated by flux balances obtained at each previous step.
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.

$$\frac{\partial}{\partial t}(\rho\bar{u}) + \nabla.(\rho\bar{u}\bar{u}) = -\nabla p + \nabla[\mu(\nabla\bar{u}^T)] + \rho g + F$$
………………… (1)

## 5. Results and Analysis

Validation of the numerical model was carried out against the data taken from the resources of dam spillway. The experimental data taken from sources was then tabulated to calculate different parameters. The validation of the experimental data included Froude number, average velocity distribution near the gate bottom by changing the gate openings at various water head.

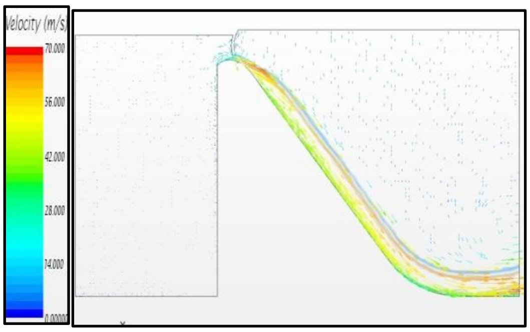


**Fig. 2**Velocity Distribution over radial gated ogee crested spillway

Fig. 2 showed the velocity distribution over radial gated ogee crested spillway for 1m of gate opening. In this the flow velocities near the gate bottom increases but as we succeeded towards end of chute velocities tend to a constant value within the uniform flow region.

### 4.1 Validation of the average velocity variation

The experimental data of the velocities at various gate openings is validated against the VOF model which gives the close agreement. Fig. 3 showed a comparison of the experimental data and the numerical model as described in the Table 1.
As seen in Fig. 3there is excellent agreement between the predicted and measured average velocities 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.

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

| Gate Opening (m) | Average Velocity (m/s) | | Difference (%) |
|---|---|---|---|
| | Experimental | VOF | |
| 0.5 | 10 | 10.955 | 8.7 |
| 1 | 47 | 46.28878 | 1.5 |
| 2 | 42.5 | 43.36421 | 2 |
| 4 | 37.5 | 38.93685 | 3.7 |
| 6 | 17.7 | 18.5856 | 4.8 |

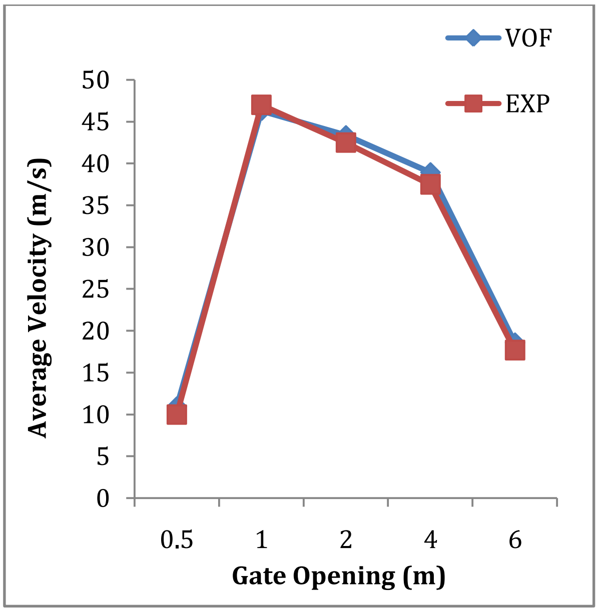


**Fig. 3**Average velocity variation at different gate openings

### 4.2 Validation of the Froude number variation

For the ogee crested spillway the flow can be classified as supercritical, critical and subcritical depending on the value of Froude no. The Froude Number was determined from the experimental velocities at each gate opening.

$$\text{Froude No. (Fr)} = \frac{V}{\sqrt{gL_c}}$$
…………….. (2)

Where, V is Average fluid velocity at cross section and $L_c$ is the characteristics length. For this problem it is taken as a gate opening height.

For this problem the Fig. 4 showed the better convergence between the experimental and VOF model.
The flow at ogee crest always supercritical for all openings as described in the Table 2. The difference between the numerical model and VOF model had never exceeded 10%.

**Table 2** Froude Number comparisonnear gate bottom of gated spillway flow

| Gate Opening (m) | Froude No. | | Difference (%) |
|---|---|---|---|
| | Experimental | VOF | |
| 0.5 | 4.52 | 4.946 | 9.55 |
| 1 | 15 | 14.779 | 1.51 |
| 2 | 9.6 | 9.79 | 2.03 |
| 4 | 6 | 6.22 | 3.83 |
| 6 | 2.31 | 2.423 | 5.0 |

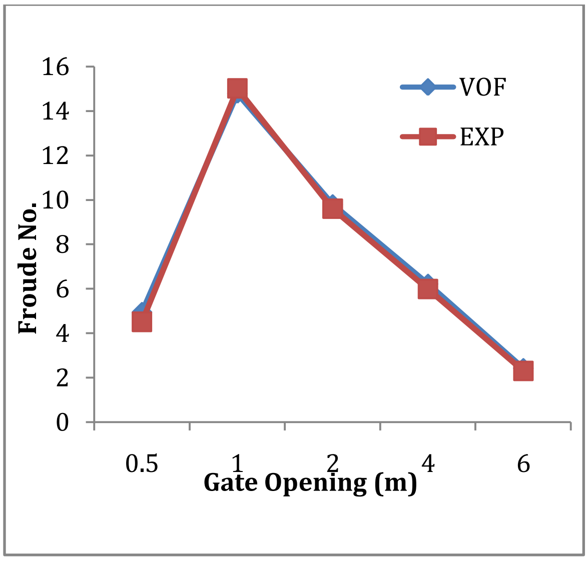


**Fig. 4**Froude No. variation at different gate openings

## 6. Conclusion

The numerical model for one of reservoir developed using STAR CCM+ CFD tool gives better agreement between the numerical model data and physical data recorded at dam reservoir.
A numerical model using VOF multiphase flow model together with RNG k-ε turbulence model was used to simulate the flow over an ogee crested spillway with gated flow. The data obtained from large scale experiments of dam reservoir was used to verify the model.
The validation of the numerical model against the measured average velocities near the gate bottom at different gate openings for the gated flow over spillway showed excellent agreement.
Also the Froude Number data calculated from the experimental parameter gives the better agreement between numerical and experimental data.
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 6m 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 parameters associated with dam flow. Finally the numerical model has many advantages in practice, in terms of parametric study.

## 7. References

FatemaZandiGoharrizi, Mehdi AzhdaryMoghadam,(2010) Evaluation of a Numerical Modeling for Flow over an OGEE Spillway, *8th International River Engineering Conference ShahidChamranUniversity*,pp. 110-116.
Dan Gessler, (2005) CFD modeling of spillway performance, *Journal on impact of global climate change*, pp. 1-10.
Fernando Salazar a , Rafael Morán b , Riccardo Rossi c d & Eugenio Oñate,( 2013) Analysis of the discharge capacity of radial-gated spillways using CFD and ANN – Oliana Dam case study, *Journal of Hydraulic Research,*1, pp. 1-9.
Riyadh Zuhair Al Zubaidy and Shaymaa Abdul MuttalebHashimAlhashimi,( 2013) Numerical simulation of two-phase flow over mandali dam ogee spillway, *International journal of structural and civil engineering research,*2, pp.1-13.
DaeGeun Kim and Jae Hyun Park, (2005) Analysis of Flow Structure over Ogee-Spillway in Consideration of Scale and Roughness Effects by Using CFD Model, *KSCE Journal of Civil Engineerin,* 9, pp. 161.
Paul G. Chanel and John C. Doering, (2008) Assessment of spillway modeling using computational fluid dynamics, *Canadian journal of Civil engineering*, 35, pp.1481–1485.
Dan Gessler (2005), CFD Modeling of Spillway Performance, *Journal on Impacts of Global Climate Change,* pp.1-10.
Jean Chatila, MazenTabbara,(2004) Computational modeling of flow over an ogee spillway, *Journal of Computers and Structures*,82, pp.1805-1812.
FatemaZandiGoharrizi, Mehdi AzhdaryMoghadam(2010), Evaluation of a Numerical Modeling for Flow over an OGEE Spillway, *8th International River Engineering Conference,* pp. 1-10.
Bruce M. Savage and Michael C. Johnson, (2001) Flow over ogee spillway: physical and numerical model study, *Journal of Hydraulic Engineering*, 127, pp. 640-649.
Robert F. Einhellig, (1999) Hydraulic Model Study of Folsom Dam Spillway Performance and Stilling Basin Abrasion, *U.S. Department of The Interior Bureau of Reclamation,pp.*125-131.
James A. Higgs, (1997) Folsom Dam Spillway Vortices Computational Fluid Dynamic Model Study, *U.S. Department of The Interior Bureau of Reclamation,* pp.31-37.

M R Bhajantri, T I Eldhoand P B Deolalikar, (2006) Hydrodynamic modelling of flow over a spillway using a two-dimensional finite volume-based numerical model, *Sadhana*, 31, pp. 743-754.
D.K.H. Ho, K.M. Boyes and S.M. Donohoo, (2001) Investigation of Spillway Behavior under Increased Maximum Flood by Computational Fluid Dynamics Technique, *14th Australasian Fluid Mechanics Conference, Adelaide University*, pp.10-14.
SadeghDehdar-behbahani, Abbas Parsaie,(2016) Numerical modeling of flow pattern in dam spillway's guide wall. Case study: Balaroud dam, Iran, *Alexandria Engineering Journal,* 55, pp. 467–473.
SebastienErpicum, YannPeltier, Benjamin Dewals, Pierre Archambeau, Michel Pirotton, (2017)Pressure and velocity on an ogee spillway crest operating at high head ratio: Experimental measurements and validation, *Journal of Hydro-environment Research,*pp.1-9.