International Journal of Science and Research (IJSR)
ISSN (Online): 2319-7064
Index Copernicus Value (2013): 6.14 | Impact Factor (2013): 4.438

Numerical Modelling of Two Tier Spillway

Ravindra R. Bhate[1], **Dr. G.A. Hinge**[2]

[1]Sinhgad College of Engineering, Vadgaon Bk, Pune 411 041, India

[2]B.S. College of Engineering and Research, Narhe, Pune – 411 041, India

Abstract: ***Spillway flows are classical problem of gravity driven free surface flow hydraulics. These flows are very complex because they are highly turbulent and unsteady involving two phases (air & water) with a large density difference. These features pose a unique challenge to simulate these flows numerically. The two tier spillway in a combination of two spillways, an overflow spillway and a sluice spillway at bottom. A study was completed to compute flow parameters over a two tier spillway using a 2-D numerical model. Water surface profile, discharge and pressure data were recorded for different flow conditions. A commercially available computational fluid dynamics (CFD) program, which solves the Reynolds-averaged Navier-Stokes equations, was used to model the two tier spillway. The results were compared with available experimental data from 1:55 scale 2-D sectional physical model to assess the accuracy of numerical results and found in good agreement.***

Keywords: Two tier spillway; CFD; discharging capacity; water profile; pressure profile.

1. Introduction

The construction of a dam is necessary for forming the reservoirs which can be used in many ways. In many cases, to allow the water simply to overtop the dam would result in a catastrophic failure of the structure. For this reason, carefully designed overflow passages - known as 'spillways' are incorporated as part of the dam design. Spillways form most important part of the dam. The spillway capacity must be sufficient to accommodate the 'largest' flood discharge likely to occur in the life of the dam. Because of the high velocities of flow often attained on spillways, there is usually some form of energy dissipation and scour prevention system at the base of the spillway. In the design of overflow spillways, information regarding the hydraulics of the flow over and around the structure is of interest. The desired hydraulic data includes discharge rating curve, pressures over spillway surface, water surface profiles, and velocity profiles. Obtaining an accurate estimate of the discharge rating curve is important as knowledge of the structure's discharge capacity allows for evaluating the capability of the spillway to safely pass the design flood at the prescribed reservoir water level. If the reservoir water levels exceed design water levels, pressures over the spillway crest can become negative. If these pressures become too low and the cavitation index drops below 0.2, cavitation may occur and cause significant damage to the concrete surface of the spillway. Water surface profiles are also often desired in order to determine appropriate heights of training and divide wall such that overtopping does not occur.

Traditionally, scaled physical models have been constructed in hydraulic laboratories to study this behavior of spillways, however they are expensive, time-consuming and there are many difficulties associated with scaling effects. Today, with the advancement in computer technology and more efficient Computational Fluid Dynamics (CFD) codes, the behavior of flow over spillway can be investigated numerically in reasonable time and expense [7]. The main attraction in using CFD, resides in its ability to investigate physical fluid systems and provide a large amount of data more cost effectively with more flexibility and rapidly than with experimental procedures. CFD is able to overcome many difficulties that the physical models encounter to measure flow quantities and phenomenon in inaccessible flow regions or due to disturbances caused by the instrument and/or by the experimental environment. CFD technique has been applied to investigate several spillway structures in Australia [4]. However, validation of the numerical model results with the results of physical model studies is essential, before it can be used for parametric studies.

2. Numerical Flow Analysis

Computational Fluid Dynamics, commonly known by the acronym 'CFD', is a branch of Fluid Mechanics that resolves numerically, fluid flow problems. The fluid motion is governed by three basic principles namely conservation of mass, conservation of momentum and conservation of energy. The physical laws governing a fluid flow problem are represented by a system of partial differential equations regrouping the continuity equation, the Navier-Stokes equations and any additional conservation equations [8].

Continuty equation

$$\frac{\partial\rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0 \qquad (1)$$

$$\rho\frac{\partial u}{\partial t}+\rho u\frac{\partial u}{\partial x}+\rho v\frac{\partial u}{\partial y}+\rho w\frac{\partial u}{\partial z}=-\frac{\partial\hat{p}}{\partial x}+\mu\left[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right] \qquad (2)$$

$$\rho\frac{\partial v}{\partial t}+\rho u\frac{\partial v}{\partial x}+\rho v\frac{\partial v}{\partial y}+\rho w\frac{\partial v}{\partial z}=-\frac{\partial\hat{p}}{\partial y}+\mu\left[\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right] \qquad (3)$$

$$\rho\frac{\partial w}{\partial t}+\rho u\frac{\partial w}{\partial x}+\rho v\frac{\partial w}{\partial y}+\rho w\frac{\partial w}{\partial z}=-\frac{\partial\hat{p}}{\partial z}+\mu\left[\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right] \qquad (4)$$

The numerical analysis resolves these equations by accurate and complex numerical schemes. A program or code, where the numerical algorithm is implemented, is then solved on a computer. This technique of replacing the differential equation governing the fluid flow, with a set of algebraic equations is known as discretization. The well known discretization methods used in CFD are Finite Difference Method (FDM), Finite Volume Method (FVM), Finite Element Method (FEM), and Boundary Element Method

(BEM). Nowadays, most of CFD codes use parallel computation in order to resolve a flow problem faster by 'sharing' the calculation and the memory required among several computers. As the performance-to-cost ratio of computers has increased at a spectacular rate in the last decade and shows no sign of slowing down, CFD is considered more often as a key industrial tool.

The challenge in solving flow numerically over a spillway is the presence of a free surface which is transient in nature. This is especially difficult when the water surface is rapidly changing with a high degree of curvature, such as when the flow changes from subcritical flow to supercritical flow and back again. It is important to track the free surface accurately to solve the flow numerically over the spillway because large streamlines curves exert non-hydrostatics pressure distribution over the section. Tracking involves, locating the surface, defining the surface as a sharp interface between water and air and applying the boundary condition at the interface. There are different means for tracking the free-surface boundary conditions. Volume of fluid (VOF) is one of them [5].

The computation becomes more complex when a turbulence model is introduced. Whenever turbulence is present in the flow it appears to be dominant over all other flow phenomena. That is why successful modelling of turbulence greatly increases the quality of numerical simulations. One of the main characteristics of turbulent flow is fluctuating velocity fields. These fluctuations cause mixing of transported quantities like momentum, energy and species concentration and thereby also fluctuations in the transported quantities. Because of the small scales and high frequencies of the fluctuations they are too computationally expensive to simulate directly in practical engineering situations. Instead, the instantaneous governing equations are time-averaged to remove the small scales and the result is a set of less expensive equations containing additional unknown variables. These unknown (turbulence) variables are determined in terms of modelled variables in turbulence models. While the Navier-Stokes equations can be solved exactly for many classical problems, any attempt to model the effects of turbulence could only be achieved in a statistical approximation manner. The Reynolds-averaged Navier-Stokes equations are commonly used for this purpose. This is an expanded form of the Navier-Stokes equations that carries the Reynolds stresses terms. Most turbulence models such as mixing-length, k-ε (turbulent kinetic energy dissipation equations), etc are ways for calculating the Reynolds stresses. Selection of an appropriate model is a skill and requires experienced modeller.

3. Numerical Model Simulation

The provision of two tier spillway is first of its kind in India. The two tier spillway consists of combination of sluice spillway and overflow spillway confirming to crest equation $x^2=240y$ and $x^{1.85}=30.5y$ respectively. A commercial CFD code, Fluent is used to investigate the hydraulic characteristics in respect of discharge, pressure distribution and water levels. Design for the two tier spillway was studied on 1:55 scale 2-D sectional physical model at CWPRS, Pune [1] and the results were used to validate the CFD model. The problem is solved on Desktop computer having configuration of Dual-Core processor of speed 2.9 GHz.

A 2-D model of the spillway is studied using "FLUENT". The geometry of the spillway is prepared using "AutoCAD" and "GAMBIT" software. The spillway domain is extended up to 50m upstream for development of approach flow and 200 m downstream of spillway to accommodate the tail water conditions. A weir is provided 150 m downstream of stilling basin end-sill so that the tail water level can be adjusted according gauge curve. Domain height is chosen around 70 m above the crest of the sluice spillway so that the water level can be attained in the reservoir as well as interface with air can be captured properly. The cells have been clustered near the sluice roof profile and spillway surface to capture wall bounded effects and predict the wall pressures in the flow simulation. The grid cell of size 0.5 m is used for meshing the domain. The Quadrilateral cells are used for grid generation with the cell count of the order of about 2.5 lakh.

3.1 Boundary Conditions

To simulate a given flow, it is important that the boundary conditions accurately represent what is physically occurring. The pressure inlet boundary condition is used at domain inlet and other air boundaries. The water outflow at the end of domain was defined as a pressure outlet boundary condition. All the solid boundaries including, sluice spillway surface, overflow spillway surface were defined as wall boundaries with no slip condition.

The VOF method was used to capture the interface between water and air and governing equations are solved by the Finite Volume Method. The k– ε turbulence model is used to simulate the two-dimensional turbulent flow[3]. The problem was started with unsteady free surface calculations with flexible time step. The steady state is reached after about 350 seconds as shown in Figure 1.

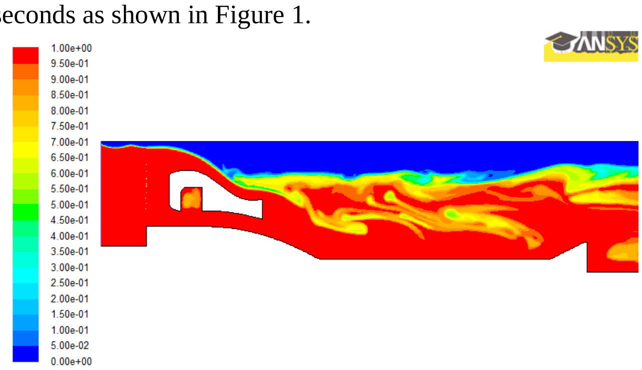
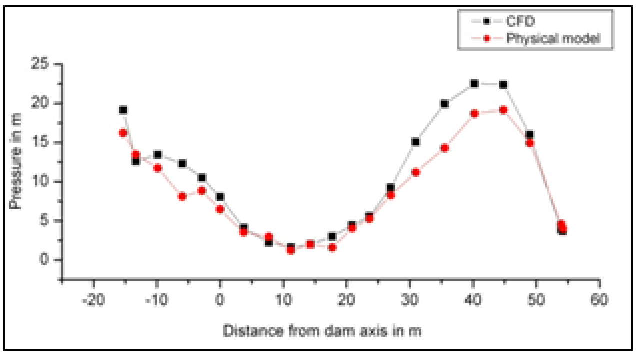


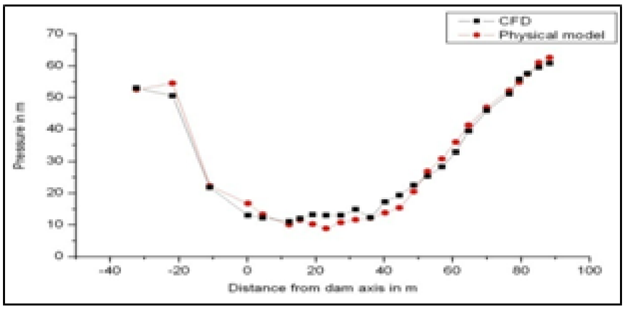
Figure 1: Phase diagram for the flow simulation over two tier spillway

The results are then computed at the various locations where the experimental data was available from the physical model study. The values observed in numerical model compare with the experimental data. The coefficient of discharge worked out for both the spillways working independently. For overflow spillway it is found around 0.61 and for sluice spillway it is found to be 0.80, which shows very good agreement with cd that found from physical model experimental studies. The trend of water surface profile and pressure distribution over the spillway profile are also matching with the experimental values. Figures 2, 3, 4 and 5 shows the comparison between the physical and numerical model for overflow spillway and sluice spillway in respect of pressure profiles for following conditions:

1) Overflow and Sluice Spillway when both spillways operating freely for PMF
2) Overflow gated and Sluice Spillway operating freely for discharge 75% of PMF
3) Overflow closed and Sluice Spillway operating freely for discharge 50% of PMF
4) Sluice Spillway operating partially for discharge 25% of PMF

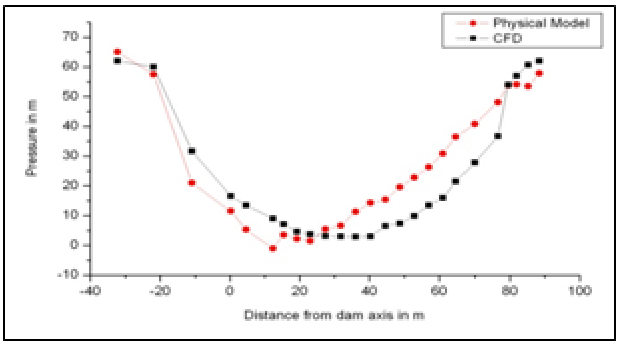


(a) Overflow Spillway

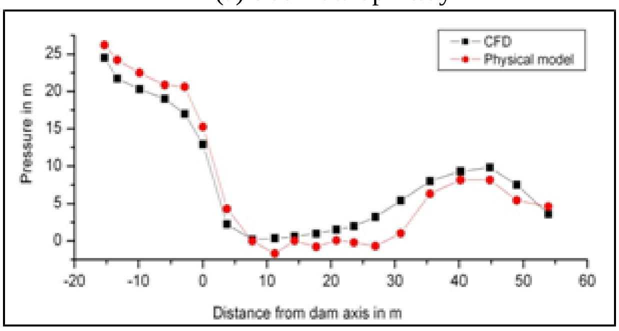


(b) Sluice Spillway

Figure 2: Pressure Profile for Condition 1



(a) Overflow Spillway



(b) Sluice Spillway

Figure 3: Pressure Profile for Condition 2

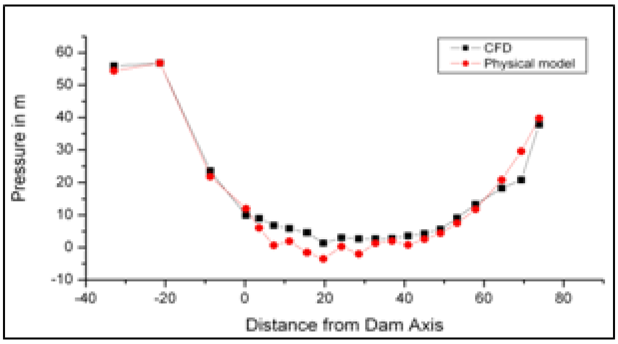


Figure 4: Pressure Profile for Condition 3

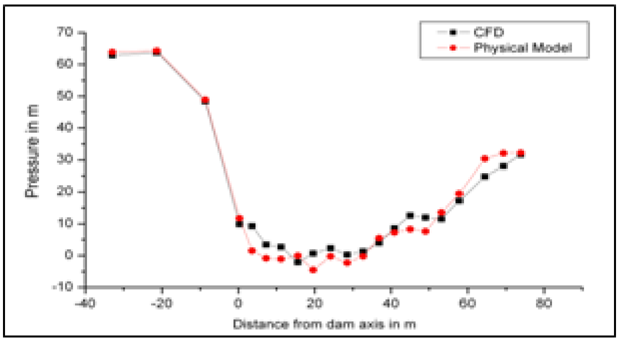


Figure 5: Pressure Profile for Condition 4

Figure 6, 7 and 8 shows the comparison between the physical and numerical model for overflow spillway and sluice spillway in respect of water profiles for following conditions:

1) Overflow and Sluice Spillway when both spillways operating freely for PMF
2) Overflow gated and Sluice Spillway operating freely for discharge 75% of PMF
3) Overflow closed and Sluice Spillway operating freely for discharge 50% of PMF

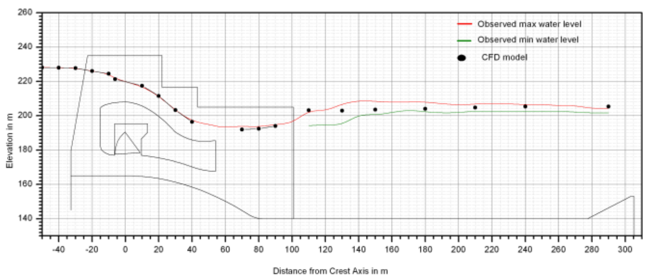


Figure 6: Water Profile for Condition 1

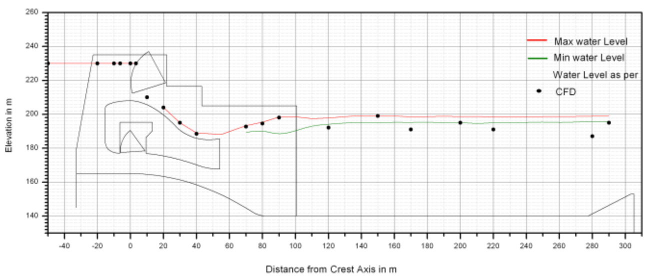


Figure 7: Water Profile for Condition 2

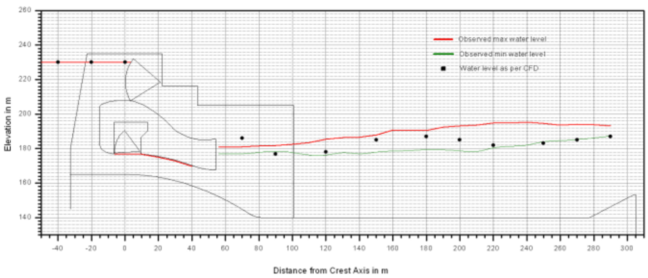


Figure 8: Water Profile for Condition 3

The figure 9 shows the comparison between physical and numerical model for discharging capacity of the spillways.

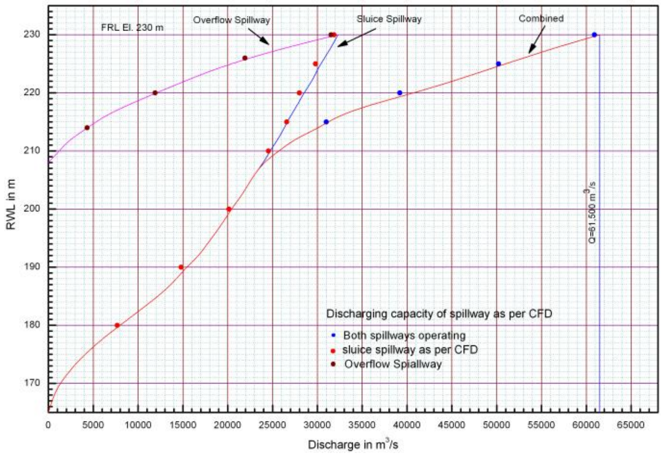


Figure 9: Discharging Capacity Curve

4. Conclusion

The flow over spillways were studied on physical model traditionally. This study shows that, numerical tools like CFD codes are also quite convenient to calculate the discharge, water and pressure profile over the spillway. The analysis for pressure profile shows 2% to 100% relative difference in the physical and numerical model readings. The high relative difference was due to very low pressure magnitude where small difference makes huge relative difference. For discharges and water profile the relative difference is of the order of 1% to 5 %. CFD gives an insight into flow patterns that are difficult, expensive or impossible to study using traditional physical modelling techniques. Although physical model studies may be more expensive and time consuming than computational modeling, they are still crucial for providing data for numerical model calibration and validation studies. The unique combination of computational expertise in physical flow modeling can be applied in concerned to provide cost effective, practical solutions to spillway flow problems. Thus, CFD modeling can be used as a complementary tool along with physical modeling to solve complex flow problems of spillways.

5. Acknowledgement

The authors are thankful to Director, CW&PRS, Pune and Dr.(Mrs). V.V.Bhosekar, Joint Director, CW&PRS, Pune for their constant encouragement and valuable guidance in preparation of this paper.

References

[1] CWPRS Technical Report No 4933 (2012), *Hydraulic Model Studies for Lower Siang H.E. Project, Arunachal Pradesh*

[2] Dargahi B. (2006), Experimental *Study and 3-D Numerical Simulations for a Free-Overflow Spillway. Journal of Hydraulic Engineering, ASCE 132-9,899-907*

[3] Gadge P, Kulhare A, Bhasokar (2011), *Application of Computational Fluid Dynamics in Hydraulic Structures, Hydro 2011*

[4] Ho, Riddette (2010), *Application of Computational Fluid Dynamics to Evaluate Hydraulic Performance of Spillways in Australia. Australian journal of Civil Engineering, Vol 6, Pg 81- 103*

[5] Ho, H., Boyes, K., Donohoo, S., and Cooper, B. (2003), *Numerical Flow Analysis for Spillways. Proc, 43rd ANCOLD Conf. Hobart, Tasmania, 24-29.*

[6] Panel, P. G., and Doering, J. C. (2007), *An Evaluation of the Computational Fluid Dynamics for Spillway Modeling. 16th Australian Fluid Mechanics Conference, 1201-1206.*

[7] Savage, B. M., and Johnson, M. C. (2001), *Flow over Ogee Spillway: Physical and Numerical Model Case Study. International Journal of Hydraulic Engineering, ASCE 127-8, 640-649.*

[8] Anderson, *CDF-The Basic with applications, McGraw Hill Inc.,New York, 1995.*

Author Profile

Shri R. R. Bhate graduated in Civil Engineering in 1998 and completed his Masters in Engineering (Civil-Hydraulics) in 2012 from Sinhgad college of Engineering, University of Pune. He joined Central Water and Power Research Station (CW&PRS), Pune in 1997 and now working as Scientist 'B' in Spillways and Energy Dissipators Division. He has more than 15 years of experience in hydraulic modelling of Spillways and Energy Dissipators. He has conducted hydraulic model studies for projects like Subansiri Lower, Lower Siang, Punatsangcchu-I, Dhauliganga, Sardar Sarovar, Kotlibhel-II, Chamera-II and III, Omkareshwar, Salma Dam etc. He is a Life Member of Indian Society for Hydraulics.

Prof. G.A. Hinge graduated in Civil Engineering from University of Pune, in 1997, Masters in Civil-Hydraulics in 2003 and Ph.D. from University of Pune in 2013. Presently, he is working as Professor (Civil Engg) in B.V. college of engineering, Pune. He has more than 15 years of experience in Teaching. He has one patent and more than 18 publications on his name. He is review panel member of international journal "Water Science and Technology" and Editorial Board Member of "Greener journal of Science, Engineering and Technology Research". He is a Life Member of Indian Society for Hydraulics.