International Symposium on Hydraulic Structures

Oct 27th, 12:00 AM

Flow Over Orifice Spillway: Physical and Numerical Study for Spillway Profile Design of Hydroelectric Project

Sushma Vyas
Central Water and Power Research Station, vyassush13@gmail.com

Y. N. Srivastava
Central Water and Power Research Station

Recommended Citation

Vyas S., and Srivastava, Y.N. (2022). "Flow Over Orifice Spillway: Physical and Numerical Study for Spillway Profile Design of Hydroelectric Project" in "9th IAHR International Symposium on Hydraulic Structures (9th ISHS)". *Proceedings of the 9th IAHR International Symposium on Hydraulic Structures – 9th ISHS*, 24-27 October 2022, IIT Roorkee, Roorkee, India. Palermo, Ahmad, Crookston, and Erpicum Editors. Utah State University, Logan, Utah, USA, 9 pages (DOI: 10.26077/66b9-64d9) (ISBN 978-1-958416-07-5).

ISBN: 978-1-958416-07-5 DOI: https://doi.org/10.26077/66b9-64d9

Flow Over Orifice Spillway: Physical and Numerical Study for Spillway Profile Design of Hydroelectric Project

Sushma Vyas[1] & Y. N. Srivastava[2]
[1]Scientist C, Central water and power research station, Pune, India
[2]Additional Director, Central water and power research station, Pune, India
E-mail: vyassush13@gmail.com

Abstract: *Many high dams on the upper Himalayan rivers which have very steep bed slopes are constructed with orifice type of Spillways with low level sluices for flood disposal and flushing of sediments. To assess the performance of spillway physical model studies were conducted on a 1:50 scale 2-D sectional model. Physical model studies can have a wide range of outcomes depending on the purposes of the model study, although many physical models frequently analyse properties such as velocity patterns, discharge rating curves, water surface profiles, and pressures. The physical measurements highlighted that the spillway is subjected to negative pressures. To overcome this, different spillway profile options were verified using numerical methods. For this, a commercially available computational fluid dynamics (CFD) program FLOW 3D was used which solves the Reynolds-averaged Navier-Stokes equations along with RNG turbulence closure model. Numerical modelling is quite beneficial when used by the designer in combination with the physical modelling. Numerical modelling enables designers to evaluate several possibilities until they find the hydraulically optimal solution, which can then be evaluated on a physical model, reducing the expense of physical model studies. In order to validate the numerical modelling results, the measured data from physical model studies for the original profile of spillway is used. Discharge passing through spillways, water surface profiles and pressures were used to compare the results of the physical model and the numerical model. When it was seen that there is reasonably good agreement between the physical and numerical model results, then all other alternatives were analyzed numerically and an alternative having a hydraulically optimal solution was selected to study on the physical model.*

Keywords: *CFD, Flow 3D, Numerical modelling, Physical modelling, Spillway*

1. Introduction

Dams and reservoirs effectively used to regulate river levels and flooding downstream of the dam by temporarily storing the flood volume and releasing it later. A spillway may be located either within the body of the dam or at one end of the dam or entirely away from the dam as an independent structure. Spillways must be operated in such a way that the water which was released to the downstream will not erode or damage the downstream toes of the dam. The surface of the spillway should also be designed to withstand erosion or scouring due to the very high velocities generated during the passage of a flow through the spillway.

Physical modelling is widely used to verify the spillway design and to study various operational issues due to complex flows. In physical modelling, scaling laws were used to convert model flow information into full scale prototype values. Due to latest advancements in computational fluid dynamics and hardware technology the use of numerical techniques for the analysis of flow over a spillway is now possible.

In the present study, a physical model was built to the geometrically similar scale of 1:50 based on Froude number criteria for assessing the hydraulic performance of a spillway in respect of discharging capacity of spillway, flow over spillway profiles and the vertical pressure distribution on the spillway profile. From the preliminary studies, it was observed that the spillway profile was subjected to very high negative pressures at 25 % and below opening of the spillway gate. To eliminate negative pressures, three different spillway profiles were numerically analyzed, and the best solution was chosen for physical modelling. For numerical simulations, authors selected gated conditions with 25% of PMF discharge. The results of numerical simulation were compared to those of physical modelling.

The following were the objectives of the present study:

a) To validate the physical model results with numerical model results for original design of spillway profile for 25% opening of radial gate.
b) Comparison of numerical model results for three alternative spillway profile in terms of discharge passing through spillway, water surface profile and pressure acting on spillway profile.
c) Selected alternative was fabricated and tested in physical model for 25% opening of radial gate of spillway.

d) Validation of the physical model results with numerical model results for selected alternative spillway profile for 25% opening of radial gate.

2. Experimental facility

The studies were conducted on a 1:50 scale two-dimensional geometrically similar physical model of a spillway based on Froude number similarity located in the Union Territory of Jammu and Kashmir. The project envisages construction of four orifice spillways and two overflow crest spillways. The orifice spillway consists of 4 spans with an elliptical bottom profile of breast walls. The energy dissipator is provided in the form of a ski jump bucket with a preformed plunge pool. Figure 1 (a) shows the side view of the dry model of spillway provided with a preformed plunge pool. The model includes a part of the upstream river, orifice spillway, ski jump bucket profile and downstream plunge pool. The model was fabricated using transparent Acrylic and PVC foam sheets. Piezometers were provided along the centre line of the centre span of the spillway for piezometric pressure measurement. Sharp crested Rehbock weir was used for measurement of discharge. Figure 1(b) shows a longitudinal section of the spillway for the original and three alternatives of spillway profile.

Figure 1 (a). Side view of the dry physical model (1:50 scale) of spillway and plunge pool for original design of spillway.

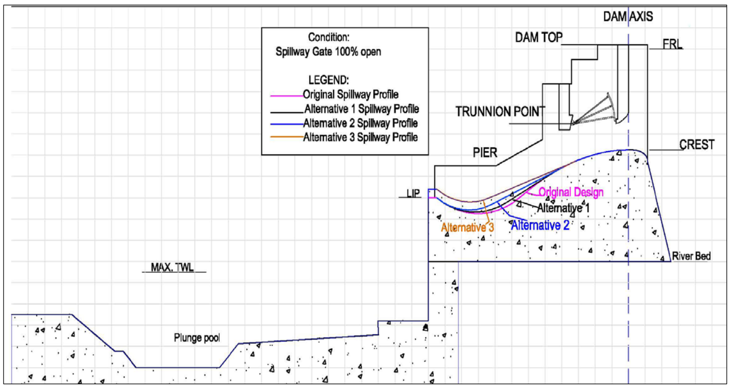

Figure 1 (b). A longitudinal section of the Prototype spillway for the original and three alternate spillway profiles.

3. Numerical methodology

Computational fluid dynamics (CFD) program Flow 3D is widely used to analyze spillway flows. The program solves the Reynolds Averaged Navier-Stokes (RANS) equations along with RNG turbulence closure model and computes free surface through volume of fluid method to track free surface motion given by Hirt and Nichols (1981). The fractional area/volume obstacle representation (FAVOR) technique given by Hirt and Sicilian (1985) is used to determine the void or flow region within each cell. One means of tracking the free surface is the VOF method. High resolution in non-flowing areas can be achieved using finer grid spacing and nested mesh technique. Complex flow physics such as short lengths of spillway, large variation in reservoir levels, high flow depths and wide range of Froude numbers varying from 3 to 9 are covered by FLOW 3D.

Savage and Johson (2001) simulated the flow over an ogee spillway using Flow-3D and found a reasonably good agreement between physical and numerical model results in terms of both pressures and discharges. However, the Flow-3D slightly overestimated the discharges. Chanel and Doering (2008) used FLOW-3D and concluded that CFD should not be considered a complete replacement for physical modelling. Ho et al. (2006), Ho et al. (2003a) and (2003b) concluded that the computed results from CFD over-estimated the velocity and hence underestimated the pressure distribution along the spillway. Chatila and Tabbara (2004), Dan Gessler (2005), Ho and Riddete (2010), Zohaib et al. (2015), Nguyen and Wang (2015) and Erpicum et al. (2017) validated the numerical predictions with experimental data with minor errors.

3.1. Numerical model setup

To simulate the given flow, a three dimensional geometry was created using AutoCad software of the similar scale as prototype and exported as a stereo lithographic (Stl) file and then this file was imported into CFD (Flow-3D) software for numerical analysis. The computational domain for numerical modeling is shown in Fig. 2. Two porous baffles were also included in the mesh as a means of computing the flow rate in Flow-3D software by defining them as flux surfaces and are arbitrarily placed at x=-33 m and x =75 m from dam axis. A manual multi grid method was adopted to speed up convergence to a steady-state solution. A rectangular Catersian grid type mesh with a uniform cell size in all directions was used for all mesh blocks. The mesh is extended upstream and downstream of the structure to view fluid movement. Coarse mesh (2 cm) selected for upstream and downstream sections and finer (1 cm) for spillway portion to reduce the computation time. Computational multi grid meshing with uniform cell size for numerical modelling is shown in Fig. 3.

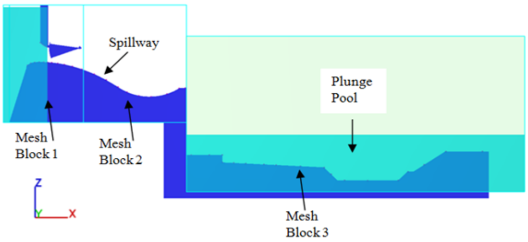

Figure 2. Computational domain for numerical modeling

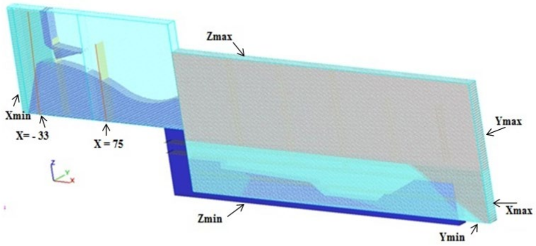

Figure 3. Computational multi grid meshing with uniform cell size for numerical modelling

To simulate given flow, it is important that the boundary conditions accurately represent what is physically occurring. Because the flow is defined in Catersian coordinates, there are six different boundaries on the computational mesh domain. To handle mesh boundary conditions, the program adds fictitious boundary cells to the grid with their values either fixed at a given value or updated as the calculations continue. The boundaries on the mesh and their coordinate directions were set as follows: Xmin – Specified pressure and fluid elevation with a hydrostatic pressure distribution; Xmax – outflow; Ymin and Ymax – wall with no slip condition; Zmin – wall with no slip condition, Zmax − pressure boundary with a gauge pressure equal to zero (atmospheric) (see figure 3). In running the Flow-3D CFD software, computation modules of viscosity, turbulence and gravity are activated for all case studies. After pre-processing, fluid flow solver option was selected. The post processing was carried out to extract the results, using the "Flow Sight" post processing software.

3.2. Validation of numerical model

In this study, prototype data was not available to compare the results of the numerical model, so we performed a validation of numerical model using the results of the physical model. Since we had already done experiments on the physical model of the original design of the spillway profile, we first did numerical modeling and validation for the same. Numerical model studies were conducted for only one span of spillway. Numerical and physical model results were compared in terms of discharge, water surface profile, and pressures along the center span of the spillway profile.

It was observed that water profile and discharge are in reasonably good agreement with each other. However, for the pressure values at few locations, a significant difference in low and high pressure values was observed, but it follows similar trend and magnitudes as observed in the physical model results (refer figure no. 4 (a) and (b)).

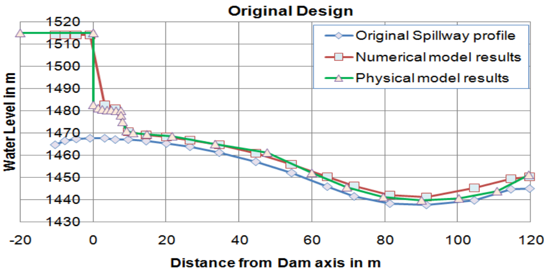

Figure 4(a). Comparison of water surface profile results from physical and numerical models for the original spillway profile

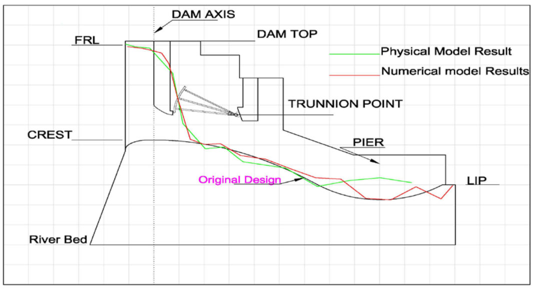

Figure 4(b). Comparison of pressures acting on spillway profile results from physical and numerical models for original spillway profile

4. Results and discussions

According to preliminary studies for the original spillway design, the spillway profile was subjected to extremely high negative pressures (ranging from - 4 m to - 9 m) at 25% and below opening of the spillway gate. Therefore, for numerical simulations, authors selected gated conditions with 25% of PMF discharge. To eliminate negative pressures and to prevent wear and tear of spillway profile due to cavitations, three alternative profiles of spillway were analysed numerically and the most effective and optimal solution was selected for physical modelling. Table 1 show the various model operating conditions considered for comparison of results.

The longitudinal section of spillway with original and all three alternative spillway profiles were shown in figure no. 5. After validating the original design, all three alternatives were numerically simulated using CFD Flow 3D software. The numerical analysis results were compared in terms of discharge, water surface profile and pressures on spillway profile. Table 1 show the various model operating conditions considered for comparison of results.

Table 1. Model operating conditions.

Spillway Profile	Operating condition in %	Required Spillway Discharge through one span in m^3/sec	Head above spillway crest in m	Froude Number (F_o)	Physical Modelling	Numerical Modelling
Original Design	25	2704	47.5	2.218	Yes	Yes
Alternative I	25	2704	47.5	2.081	No.	Yes
Alternative II	25	2704	47.5	2.191	No.	Yes
Alternative III	25	2704	47.5	2.823	Yes	Yes

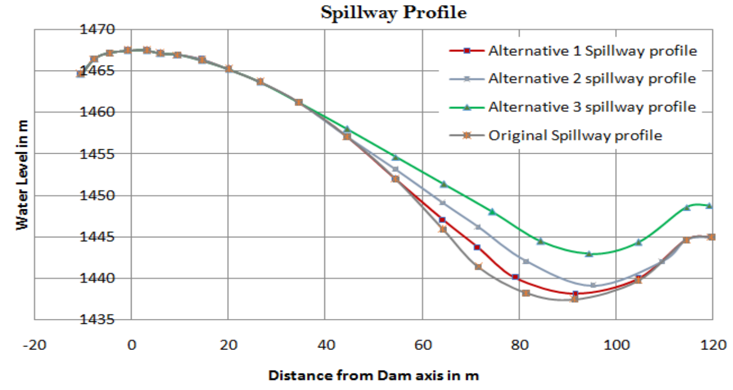

Figure 5. Longitudinal section of spillway with original and all three alternative spillway profiles

4.1. Water surface profile

Water surface profiles are important to ensure that flood water is not interfering with other structures such as gate trunnion, bridges at crest or raised gates or overtopping the spillway divide walls. From numerical analysis it is observed that for all alternatives, the water surface profiles are following spillway profile and it is not interfering with the gate trunnion or raised gates or is not overtopping the divide walls throughout the length of spillway chute (refer figure no. 6 to 9)

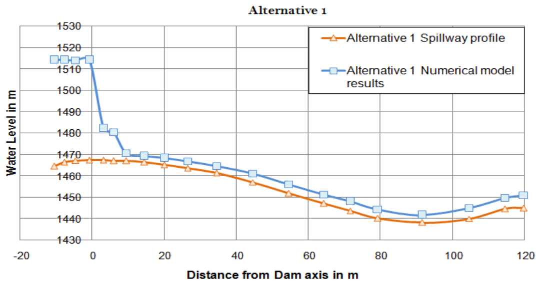

Figure 6. Plot for water surface profile for Alternative I obtained from numerical analysis.

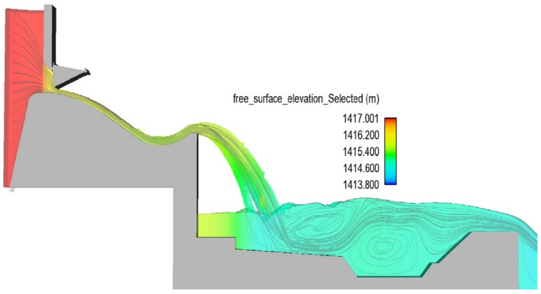

Figure 7. Water surface profile for Alternative I obtained from numerical analysis.

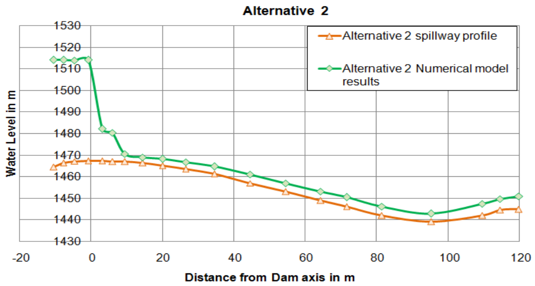

Figure 8. Water surface profile for Alternative II obtained from numerical analysis

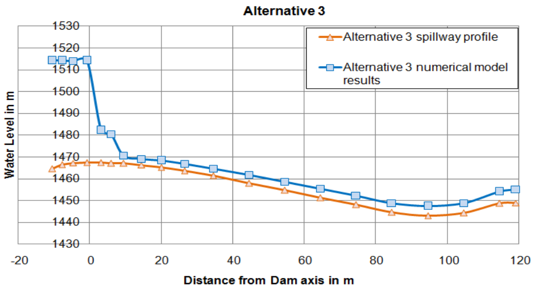

Figure 9. Water surface profile for Alternative III obtained from numerical analysis

4.2. Discharge passing through spillway opening

Table no. 2 shows a comparison between numerically obtained discharge and required discharge at 25% gate opening at reservoir level at FRL for all three alternatives. When the physical and numerical model results for the discharge passing through the spillway for a 25% gate opening were analyzed, it was found that the discharge percentage difference for the original profile was 5.31 percent, while it was 6.72 percent for Alternative 3. It indicates that the more discharge is passing through Alternative 3 profile as compared to original profile. The discharge through the Alternative III spillway profile is nearly 1.38 % more than that passes through the Alternative I spillway profile.

Table 2. Comparison of discharge passing through each alternative from numerical analysis.

Spillway profile	Operating condition	Required Model Discharge passing through spillway in m^3/s	Numerical model Discharge in m^3/s	Percentage Difference between numerical and physical model discharge
Original	Reservoir upstream water level at FRL and 25% opening of spillway gate	2,704.00	2,847.55	5.31%
Alternative 1			2846.16	5.26%
Alternative 2			2848.97	5.36%
Alternative 3			2885.72	6.72%

4.3. Pressure distribution on spillway profile

The possibility for cavitation damage caused from excessive sub atmospheric pressure was analyzed using pressure simulations along the spillway chute. Figure 10 to 13 shows the pressure distributions over spillway profile for all three alternatives for 25% opening of gate. The spillway profile was raised at a tangent point at the junction of the spillway downstream profile and bucket, to achieve positive pressure values on the spillway profile.

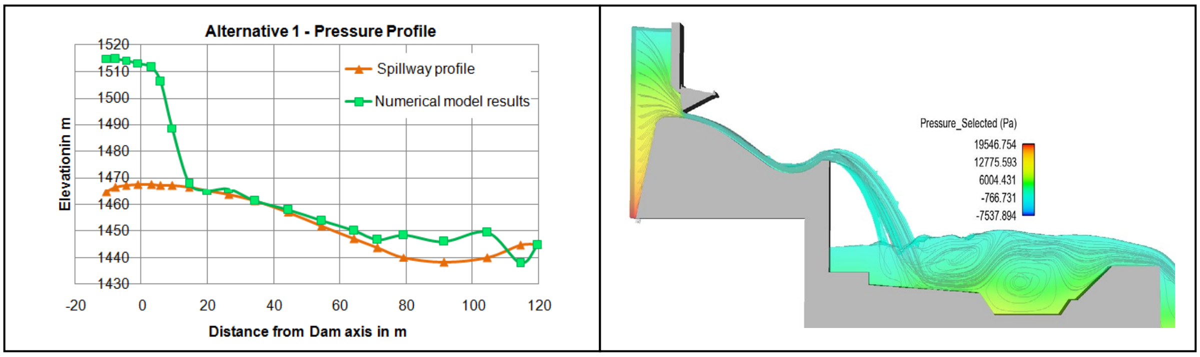

Figure 10. Pressure distribution over spillway profile for Alternative 1

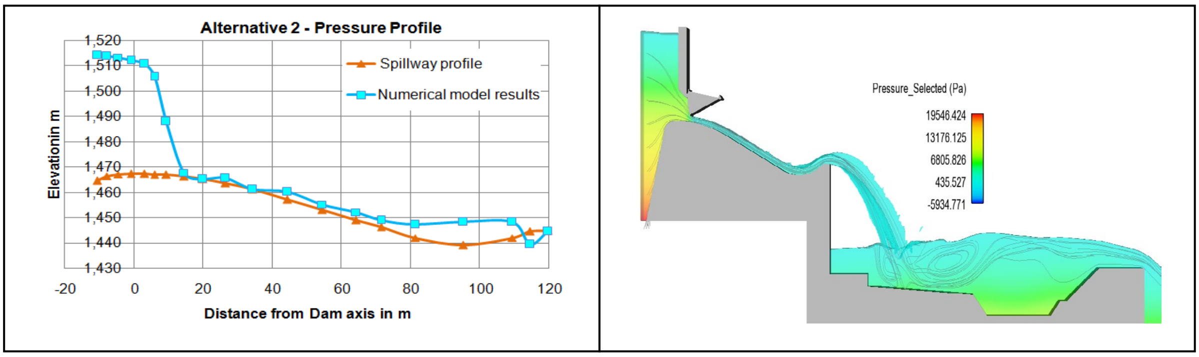

Figure 11. Pressure distribution over spillway profile for Alternative 2

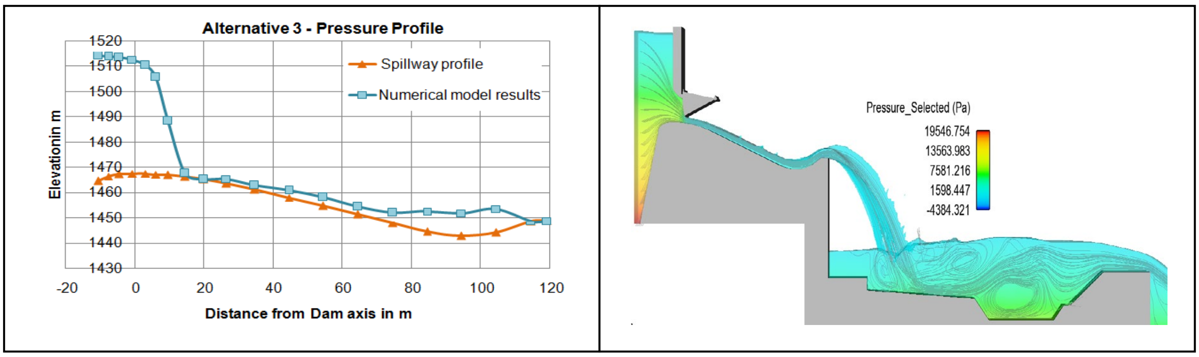

Figure 12. Pressure distribution over spillway profile for Alternative 3

In the bucket part, there is more difference in the results of the physical model and the results of the numerical model, this may be due to shorter simulation time because of which steady state is not achieved or due to coarser mesh. From Figure 10, we can see that for alternative 1, the pressure values improved due to lifting of the bucket at the tangent point. For alternative 2, the bucket is lifted further (more than alternative 1) at the tangent point, due to which the pressure values are further improved (see Figure 11). For alternative 3, the spillway profile and bucket lip both are raised. Figure 12, shows the pressure distribution over spillway profile for alternative 3.

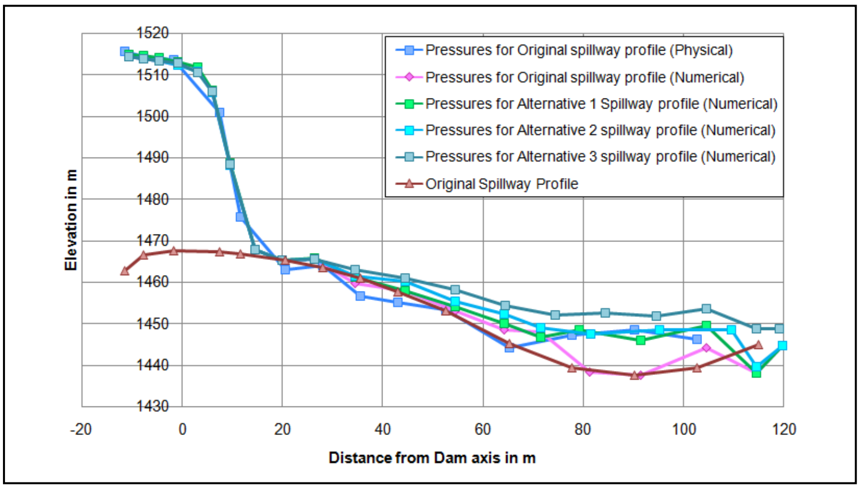

Figure 13. Comparison of pressures acting on spillway profile for all three alternatives

From figure 12, it can be seen that the pressure values are fairly positive as compared to alternative 1 and 2. Therefore, alternative 3 is selected for further study on physical model. Figure 13 shows the comparison of numerical and physical model results for original and all three alternatives.

4.4. Selection of Alternative and comparison with physical model results

After comparing numerical model results for all three alternatives, Alternative 3 was selected for physical model fabrication. After fabrication experiments for various operating conditions were repeated on a physical model, and the findings were compared once again with the numerical model results. Figure 14 shows a comparison of pressures acting on the spillway profile in a physical and numerical model. Figure 14 shows that the model predictions are in good agreement with the experimental data. There is a difference in pressure spikes at a few locations between the physical model and the numerical model results. This is due to the use of a relatively coarse mesh, which was chosen to reduce simulation time.

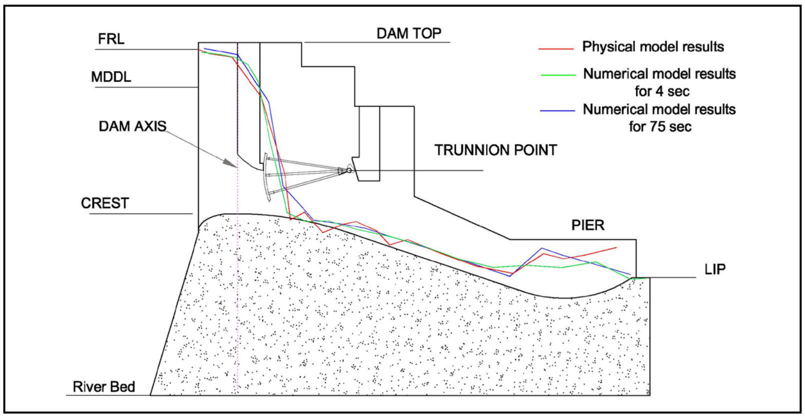

Figure 14. Comparison of results for pressures acting on the spillway profile obtained from physical and numerical model studies for Alternative 3

4.5. Comparison of physical model results for Original and selected design

The physical model results obtained for original design of spillway profile and selected Alternative 3 spillway profile were compared and it is observed that more discharge is passing through alternative 3 than through original design. The discharge passing through the Alternative 3 spillway profile is nearly 1.34 % more than that passes through the original spillway profile. From figure 15 and 16 it can be seen that the water surface profile and pressure profile are improved significantly.

From above study, it can be concluded that, the use of numerical modelling as a complement to physical modelling is extremely advantageous to obtain hydraulically optimal solutions and lowering the costs of physical model studies.

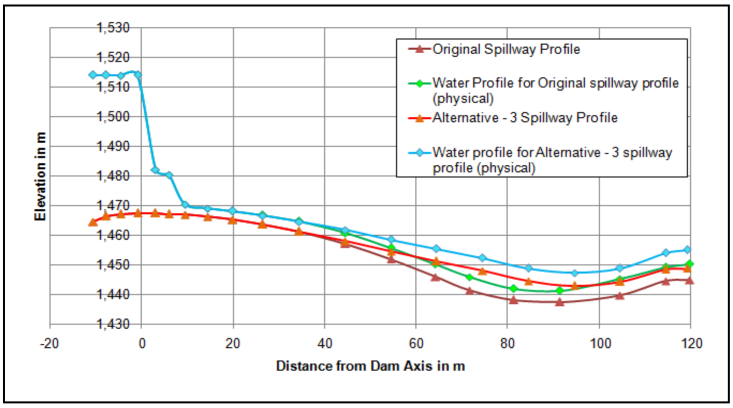

Figure 15. Comparison of water surface profile obtained from physical model results for Original and Alternative 3 spillway profile.

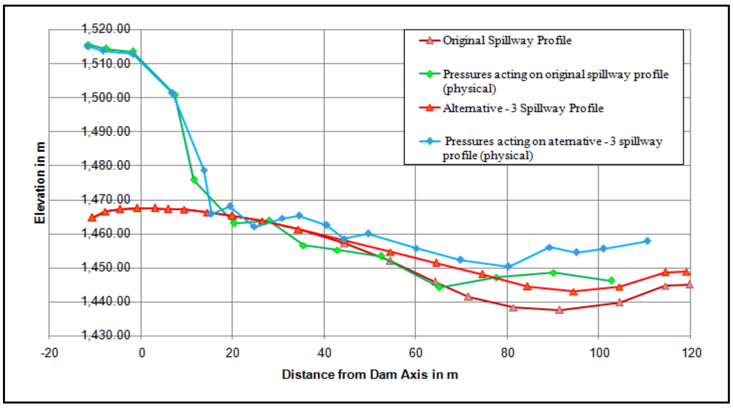

Figure 16. Comparison of pressures acting on spillway profile obtained from physical model results for Original and Alternative 3 spillway profile

5. Conclusion

This paper investigated the use of CFD Flow 3D modelling in combination with physical modelling. In the physical model study, it was observed that at 25% and lower opening of the spillway gate, the spillway profile was subjected to extremely high negative pressures (rages between (-) 4 m to (-) 9 m) for the original spillway profile. As a result, the authors chose a gated condition with a PMF discharge of 25% for numerical simulations. In order to eliminate negative pressures and prevent cavitation damage to the spillway profile, three alternative spillway profiles were numerically analysed, and the most effective and optimal solution was chosen for physical modelling.

According to the numerical analysis, the water surface profiles for all three alternatives follow the spillway profile and are not interfering with the gate trunnion or raised gates, nor are they overtopping the divide walls along the length of the spillway chute.

It is also observed that more discharge passes through alternative 3 than through alternatives 1 and 2. The discharge through the Alternative 3 spillway profile is nearly 1.34 % more than that passes through the original design of spillway profile.

The numerical simulation shows that the alternative 3 spillway profile has marginally improved negative pressures. As a result, alternative 3 was chosen and fabricated in the physical model for further studies on 2-D sectional models. The physical model results were compared to the numerical model results once more, and it was observed that the physical and numerical model results are in reasonably good agreement. Further for more refinement of results and reduce errors, we can also introduce more fine mesh, other relevant turbulence models and air entrainment models. Therefore, it can be concluded that the use of numerical modelling as a complement to the physical modelling is extremely advantageous for obtaining hydraulically optimal solutions and lowering the costs of physical model studies.

6. ACKNOWLEDGMENT

The authors are thankful to Dr. R. S. Kankara, Director, Central Water and Power Research Station, Pune for his constant encouragement and kind permission to publish the paper. The authors are also thankful to the project and design engineers for constant interaction during the course of studies.

7. REFERENCES

Chanel, P.G and Doering, J. C. (2008). "Assessment of spillway modelling using computational fluid dynamics." *Canadian Journal of Civil Engineering*, pp: 1481-1485.

Chatila, J., and Tabbara, M. (2004). "Computational Modeling of Flow over an Ogee Spillway." *Computers and Structures* 82. 1805-1812.

Gessler, D. (2005). "CFD Modelling of Spillway performance." *Proceedings of world Water and Environmental Resources Congress 2005, American Society of Civil Engineer, Anchorage, Alaska*.

Hirt, C. W and Nichols, B. D. (1981). "Volume of fluid method for dynamic of free boundaries." *Journal of Computational Physics*, Vol. 39, pp. 201-225.

Hirt, C. W. and Sicilian, J. M. (1985). "A Porosity technique for the definition of obstacles in rectangular cell mesh." *Proceedings of 4th international conference of ship dynamic*, *National Academy of Science, Washington D.C*, 1985.

Ho, D. K. H., Boyes, K. M. And Donohoo S. M. (2001). "Investigation of Spillway Behaviour under Increased Maximum Flood by Computational Fluid Dynamics Technique." *14th Australasian Fluid Mechanics Conference Adelaide University, Adelaide, Australia* 10-14 December 2001.

Ho, D. K. H. and Riddette, K. M. (2010). "Application of computational fluid dynamics to evaluate hydraulic performance of spillway in Australia." *Australia Journal of Civil Engineering*, Vol. 6, No.1, pp: 81-104.

Ho, D., Boyes, K., Donohoo, S., and Cooper, B. (2003b). "Numerical Flow Analysis for Spillways." *Proc. 43rd ANCOLD Conference, Hobart, Tasmania*, Oct 24-29, 2003: 227-237.

Ho, D. K. H., Cooper, B. W., Riddette, K. M., and Donohoo, S. M. (2006). "Application of Numerical Modelling to Spillways in Australia." *Dams and Reservoirs, Societies and Environment in the 21st Century: Proc. of the International Symposium on Dams in the Societies of the 21st Century, 22nd International Congress on Large Dams* (ICOLD), Barcelona, Spain, June 18, 2006, L. Berga et al., eds., Taylor & Francis Group, London, UK.

Ho, D. K. H., Donohoo, S. M., Boyes, K. M., and Lock, C. C. (2003a). "Numerical Analysis and the Real World: It Looks Pretty But Is It Right?" *Proc. of the NAFEMS World Congress, Orlando, FL,* May 27-31, 2003.

Thanh, N.C., Ling-Ling, W. (2015) "Physical and numerical model of flow through the spillways with a breast wall.". *KSCE J Civ Eng 19*, 2317–2324 (2015). https://doi.org/10.1007/s12205-015-0742-0

Savage, B. M. and Johnson, M. C. (2001). "Flow over ogee spillway: Physical and Numerical Model Case Study." *Journal of Hydraulic Engineering,* Vol. 127, No. 8, pp: 640-649.

Peltier, Yann & Dewals, Benjamin & Archambeau, Pierre & Pirotton, M. & Erpicum, Sébastien. (2017). "Pressure and velocity on an ogee spillway crest operating at high head ratio: Experimental measurements and validation.". *Journal of Hydro-environment Research.* 19. 10.1016/j.jher.2017.03.002.

Zohaib, N. , Sarwar, M. K. , Nabi Ghulam, (2015). "Hydraulic performance assessment of an orifice spillways using CFD modeling." *Sci. Int.(Lahore),*27(2),1303-1308.