JETIR.ORG ISSN: 2349-5162 | ESTD Year : 2014 | Monthly Issue
JOURNAL OF EMERGING TECHNOLOGIES AND INNOVATIVE RESEARCH (JETIR)
An International Scholarly Open Access, Peer-reviewed, Refereed Journal

Hydraulic design of Spillway and energy dissipator for a Run – off-the river Hydroelectric Project – comparison of Physical and Numerical model studies with a case study

[1]A. H. Gaikwad, [2]Amit Kulhare, [3]P. T. Nimbalkar,

[1]M.Tech. Student (BVDU COE), [2]Scientist 'C'(CWPRS), [3]Professor (BVDU COE)
[1]Department of Civil Engineering
[1]Bharathi Vidyapeeth (Deemed to be University)College of Engineering, Pune - 411043, India

Email: amol_gaikwad68@yahoo.com

Abstract: Spillway flows are essentially rapidly varying flows which are very complex in nature and can be studied through both physical and numerical modeling techniques. Physical modeling techniques have been extensively used in finalizing the design of spillways and energy dissipators for dams over the years and they are indispensable. However, through recent advances in computing power and modeling software capabilities, it is now feasible to undertake complex three-dimensional analysis using Numerical/Computational Fluid Dynamics (CFD) techniques. The major benefit of the CFD modeling is that it allows early identification of problematic flow features and modifications to the design/layout could be tried rapidly and cost-effectively. This paper describes the Hydraulic model studies conducted on a 1:45 scale 2-DPhysicalmodel to optimize the design of orifice spillway for a Run-off-the river Hydroelectric Project in Bhutan. Numerical model studies are carried out to assess the performance of spillway using CFD software, FLOW-3D.Various studies for parameters such as discharging capacity, pressure distribution along the spillway profile, velocities, water surface profiles over the spillway, flow conditions, were carried out using the numerical model. The results obtained from the numerical model were compared with the results of the physical model studies. Comparison of the results from the two modeling approaches has provided a very good insight into the design process to optimize the spillway design. After studying the performance of the orifice spillway, both through physical and numerical techniques, it is found that numerical/CFD modeling can be used as a complementary tool to test the preliminary design and in the optimization phase of the spillway design prior to taking up the physical model studies.

Keywords: Orifice spillway; FLOW-3D; discharging capacity; pressure profile; water surface profile

1. INTRODUCTION

Problems generally associated with complex spillway flow phenomena are identified as non-uniform flow in the approach portion creating vortices, rapidly varied flow because of complex geometry, high velocities due to high heads leading to cavitation damages, high turbulence causing hydrodynamic forces on the structure and erosion of the river bed and banks downstream, flow induced vibration for wide range of operating conditions. In such complex hydrological scenario, assessing the overall hydraulic performance of spillway would be of vital importance. Conventional 2-D/3-D physical model studies are being used extensively over the years for assessing the performance of spillways of dams for various river valley / Hydroelectric projects in order to finalize the designs. These techniques are tried and tested and the outputs from the model testing in terms of observations are invaluable in the design process. However, if the design requires modifications to optimize the design parameters in order to improve the performance of the structure, modifications on the model are required to test the alternative designs which often proves to be time consuming and cost prohibitive. Construction, operation, and testing of physical model itself will take more time and demanding lot of space thereby making it expensive. To get the initial designs tested on a physical model, the designer has to wait for a long time. In view of this now a days, Numerical approach has become very popular for its flexible, quick and economic performance.

Through recent advances in computing power and modeling software capabilities, it is now feasible to undertake complex three-dimensional analysis using Numerical /Computational Fluid Dynamics (CFD) techniques (Savage and Johnson, 2001). To date, Numerical modeling has generally been used as a valuable tool in the optimization phase of the project prior to taking up the physical model studies. The major benefit of the CFD modeling is that it allows early identification of problematic flow features and modifications to the layout could be tried rapidly and cost-effectively. Also, numerical models have the benefit of providing more insight in terms of showing the entire discretized pressure and velocity fields of the flow. Although CFD modeling packages now have the capability to analyze complex hydraulic conditions common in spillways such as air entrainment, flow separation, turbulence and shock waves, there is however a significant lack of calibration and validation studies (between CFD and physical models and also between model and prototype) for these advanced applications and caution should be applied to their use in design. As such, Numerical modeling and physical modeling are most productive when combined, because good Numerical modeling requires good physical understanding of the realistic flow processes and flow conditions. It is not possible to replace the physical models since validation of the numerical model is done using the results of physical models. Hydraulic design of spillway can be optimized by judicious combination of both physical and numerical modeling.

The present paper deals with the Hydraulic model studies conducted on a 1:45 scale 2-D Physical model to optimize the design of orifice spillway for a Run-off-the river Hydroelectric Project in Bhutan. Numerical model studies were also carried out to assess the performance of spillway using Computational fluid dynamics software, FLOW-3D. The main task of the work was to find out how well flow phenomena’s occurring in a physical model can be simulated with the numerical model/CFD software. Simulation of flow features will be mainly to investigate hydraulic aspects/parameters such as discharging capacity, pressure distribution on the spillway profile, water surface profiles, velocities and flow conditions to study the performance of the spillway and energy dissipator.The results of the numerical studies obtained using Flow 3D software are compared with results of the physical model studies to know how best they are matching and draw conclusions from both the modeling techniques.

1.1 CASE STUDY/ PROJECT DETAILS

The proposed H.E. Project with installed capacity of 600 MW is located at the lower course of river just before its confluence with Dramengchu (Gongrichu) in Trashiyangtse District of Bhutan. The area lies partly in the Khamdang Gewog (block) on the left bank and partly in the Tomi Jangsa Gewog on the right bank. The project envisages construction of 167 m long and 62 m high concrete gravity dam above river bed. The dam has been provided with a breast wall spillway (Ogee plus Sluice type) with 5 spans of 6.8 m wide x 12.9 m high in the lower tier including one bay of upper tier overflow ogee with open crest in the pier body. The sluice spillway is equipped with radial gates separated by 8.012 m thick piers at dam axis and are tapered till the end of the pier nose. The crest level of the Sluice spillway is at El. 1515 m. FRL is at El 1572 m and MDDL is at 1558 m. The downstream profile conforms to eq. X^2=202 Y and upstream profile conforms to equation X^2 /101072 + Y^2/58732=1. The spillway is required to pass PMF discharge of 8750 m^3/s. A ski-jump bucket with 40 m radius, 35º lip angle with its invert at El. 1503 m is provided for energy dissipation. A concrete apron with varying elevation downstream of the ski-jump bucket has been provided. Figures 1 shows thecross section of spillway.

2. PHYSICAL MODEL STUDIES

A 2-D sectional model was constructed to a geometrically similar scale of 1:45scale after duly incorporating all the design features as cited above. Figure 2shows physical model constructed at CWPRS. Necessary arrangements were made for measurement of discharge, reservoir water levels and pressures. The accepted equations for similitude, based on Froudian similitude were used to express the mathematical relationship between the dimensions and hydraulic parameters of the model and the prototype.

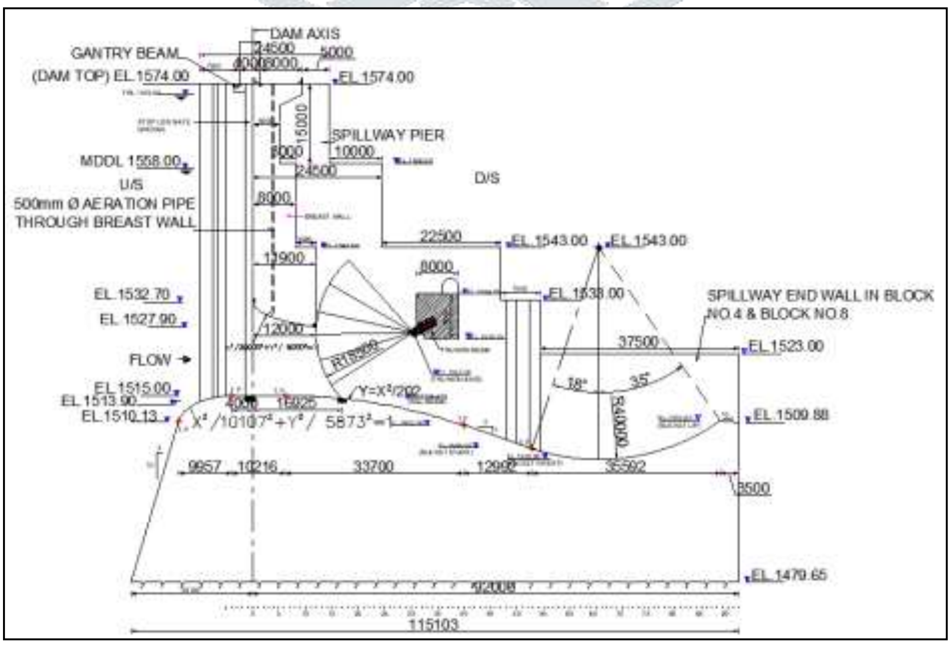

Figure 1: Cross section of spillway

Figure 2: Model the of spillway

Figure 3: Flow conditions over the spillway for a discharge of8750m^3/s, ungated operation of spillway

Hydraulic model studies were conducted to assess the performance of spillway in respect of discharging capacity, water surface profiles, velocities and pressure distribution along the bottom profile of the spillway for various operating conditions, andto assess the efficacy of energy dissipator in the form of ski-jump bucket for the entire range of discharges. Figure 3 shows the flow conditions over the spillway and ski-jump bucket for a discharge of 8750m^3/s at ungated operation of spillway.

3. NUMERICAL MODEL STUDIES

The commercial computational fluid dynamics software Flow-3D was used to simulate the flow through the Orifice spillway. The software uses a control volume-based technique to convert the governing equations in algebraic equations that can be solved numerically. The geometry of the model was prepared in “AutoCAD” software. The geometry was made with one full span and 2 full piers in 1:45 scale. The prepared geometry then exported to FLOW-3D code in the form of stereo lithography files (*.stl). To define the flow regions the mesh extended up to 2 m upstream and 4 m in the downstream side of the spillway axis. The flow region was subdivided into a mesh of fixed rectangular cells. A single mesh block can be seen in figure 4. 17 numbers of probs were provided at different locations for observing the different parameters of the flow such as pressures, velocities,free surface elevations etc. Baffles were provided at upstream and downstream locations to monitor flow parameters for convergence and steady state of the simulation.

Numerical problems are defined in terms of initial conditions and boundary conditions of the domain. When solving the Navier-Stokes equation and continuity equation, appropriate initial and boundary conditions need to be applied. The main boundary conditions in the discretized equations of the finite volume method are inlet, outlet, wall, prescribed pressure, symmetry etc.

The inlet of the domain was defined as a "specified pressure" boundary condition specifying the head of the water as reservoir water level in m head over the crest. Side and bottom of the domain was specified with the wall boundary condition. Outlet of the domain was specified as "specified pressure" boundary condition with tail water level elevation to observe the effect of ski-jump jet with tail water in the downstream of the spillway. Figure 4 shows the extent of the domain along with the different boundary conditions.

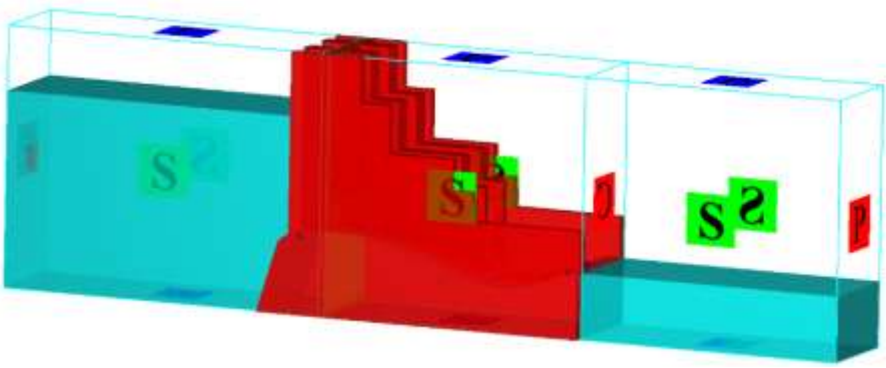

Figure 4: Numerical model showing boundary

Initial conditions define the starting conditions in the simulation. The initial state of the solution for transient fluid flow problems must be known in order to find a solution and the initial conditions are assumed, approximating the true state at time t = 0. To initialize the solution two fluid zones were created; one for reservoir zone upstream and second fluid zone for maintaining tail water downstream of the spillway structure. Apart from the basic settings for the two face flow under gravity, viscous flow with the RNG turbulence model was used to solve the governing equations for the solution.The VOF method was used to capture the interface between water and air and governing equations were solved by the Finite volume method. Various aspects in terms of flow conditions, discharging capacity, pressure distribution along bottom profile of the spillway, velocity (vectors), water surface profile was computed from the numerical model. The simulation was carried out for 80 seconds and the variation in discharge at the orifice opening and downstream of model was monitored for steady-state condition.

4. COMPARISION OF RESULTS OF PHYSICAL AND NUMERICAL MODELS

The results obtained from numerical model were compared with findings of physical model in respect of discharge, water surface profile, pressures and velocity distribution for ungated operation of spillway for design discharge 8750m^3/s. The discharge, water profile, pressure profile and velocities and flow conditions observed in the physical model are compared with the corresponding with the corresponding parameters computed from numerical model.

4.1 DISCHARGING CAPACITY OF SPILLWAY

Studies were conducted for discharging capacity of spillway with full gate opening for various reservoir water levels. The design discharge of 8750 m^3/s could be passed at RWL El 1552.33 m with all five gates fully open. It was observed that a discharge of 11,549 m^3/s could be passed at FRL El. 1572 m with all gates fully open. With one span inoperative, a discharge of 9238 m^3/s could be passed at FRL. In numerical model a discharge of 9154 m3/s has been realized RWL El 1552.33 m. So, an error 4.6% is observed in comparison of physical and mathematical model discharges. As this deviation is small and hence the discharging capacity of the spillway can be considered adequate.

4.2 WATER SURFACE PROFILES

Water surface profiles were measured on the physical model along the center line of spillway for the discharge of 8750m^3/s for ungated operation of the spillway. For numerical simulation, water surface profiles over surface of spillway were measured at 80 seconds which corresponds to the occurrence of stable flow conditions. Figures 5 and 6 show the obtained water surface profile from numerical simulation for a discharge of 8750 m^3/s, for ungated operation of the spillway.

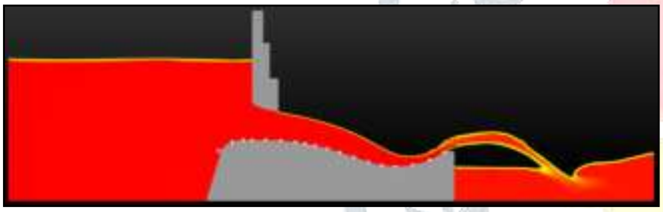

Figure 5: Water surface profile from Numerical simulation for a discharge of 8750 m^3/s (Ungated operation)

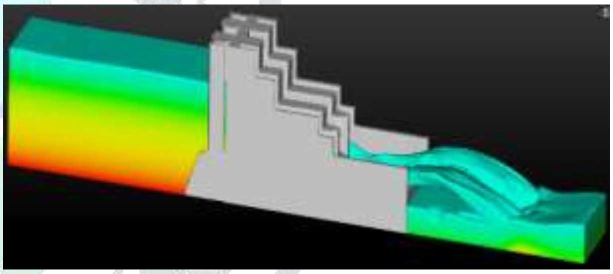

Figure 6: 3D view of water surface profile from Numerical simulation for a discharge of 8750 m^3/s (Ungated operation)

Figure 7 shows the graphical representation of comparison of observed and simulated water surface profiles for ungated operation of the spillway. It is found that the results obtained from the physical and numerical model are in close agreement throughout the length of the spillway, except at few chainages. The reasons for the error generation in the numerical model studies are primarily caused by the grid density, especially in the regions close to the interface. The errors can be reduced with a denser grid, but the computing time will be increased accordingly. Rooster tail formation in the physical model also could be reason for the deviation in the water profiles.

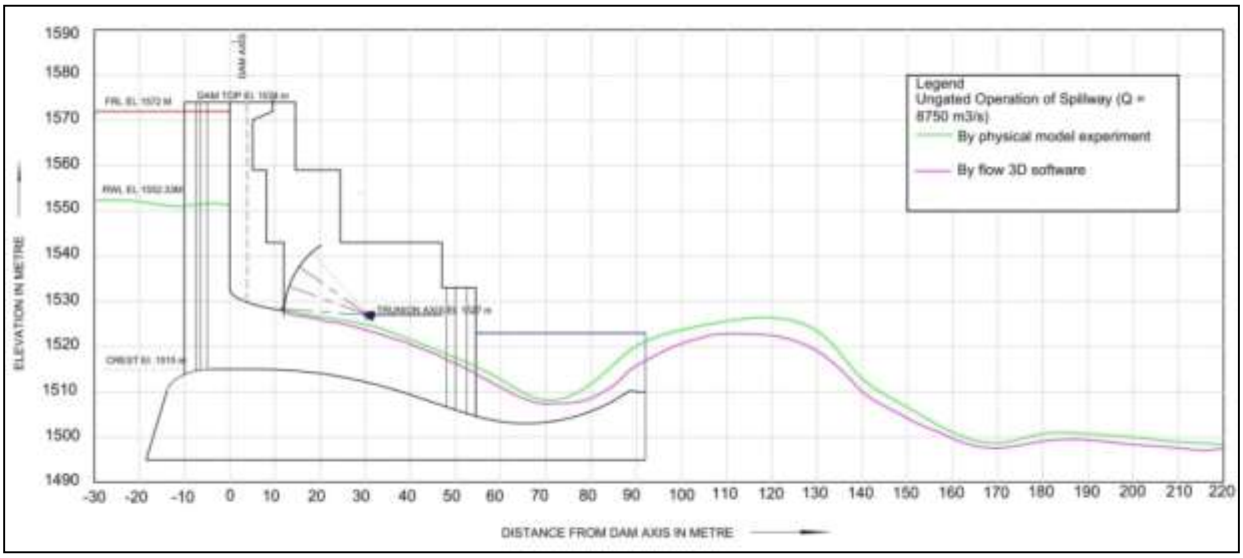

Figure 7: Comparison of Water surface profiles for the discharge of 8750 m^3/s (Ungated operation)

4.3 PRESSURE DISTRIBUTION / PROFILES

Piezometric pressures were observed at various locations over the spillway surface in metres of water column along the centre line of the spillway for various operating conditions. It was observed that piezometric pressures along the centre line throughout the profile of the spillway were positive for the entire range of 8750 m^3/s discharge with ungated operation of the Spillway. For numerical simulation, pressures at pre-defined points were measured at 60 seconds. The pressures observed on numerical model are in close agreement with the results of the physical model.Figures8 and 9 show the average mean pressure from numerical simulation for the discharge of 8750 m^3/s and comparison of pressures for the discharge of 8750 m^3/s for ungated flow.In the Figure 8 the magnitude given in the legend of pressure contours are given in Pascal and hence to be divided by ρg for obtaining pressure in meter. Where ρ is density of water and g is acceleration due to gravity.

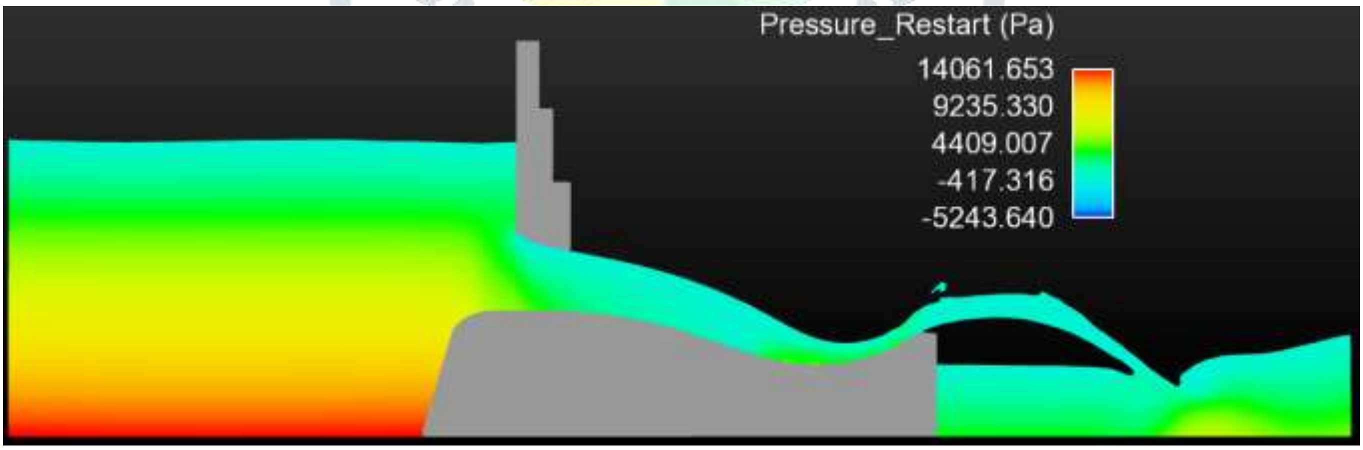

Figure 8: Average mean pressure from numerical simulation for discharge of 8750 m^3/s (Ungated operation)

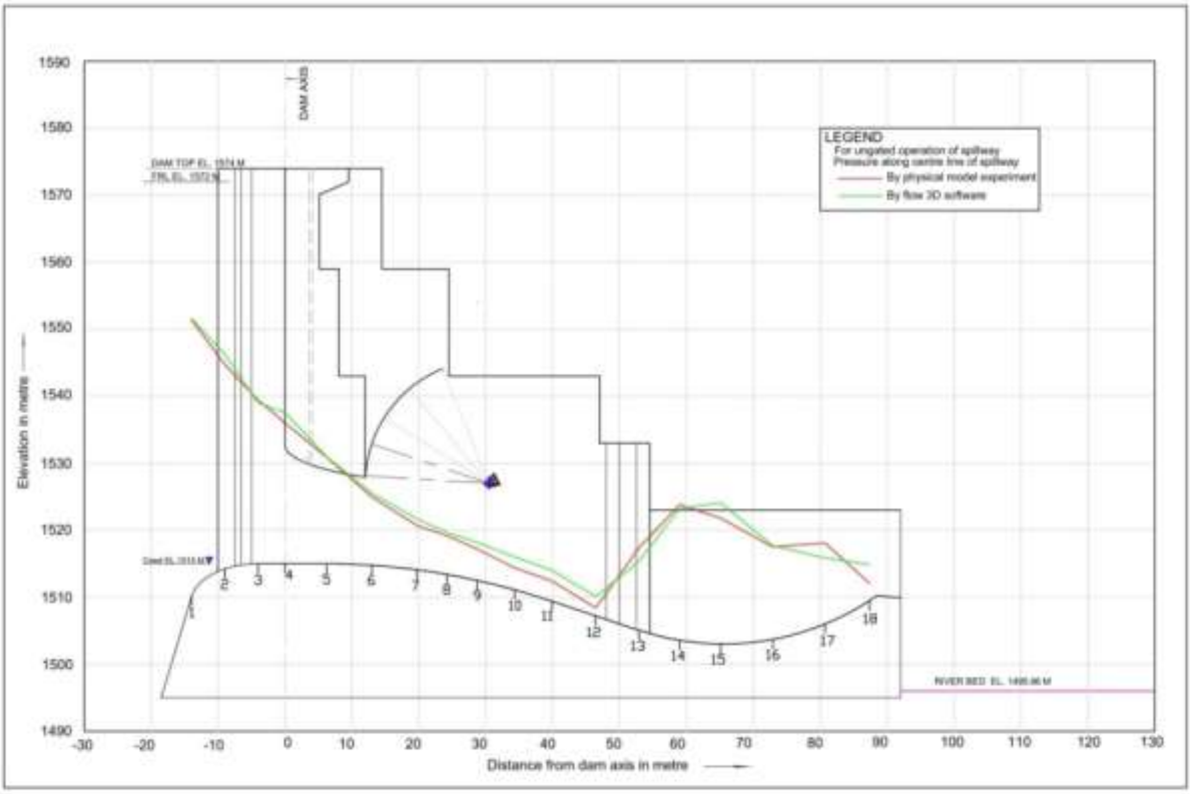

Figure 9: Comparison of pressures for a discharge of 8750 m^3/s (Ungated operation)

4.4 VELOCITY OBSERVATIONS

Velocity measurements were taken at different locations over the spillway profile on the physical model. The velocities at the crest of the spillway and at the invert, lip of the bucket were observed. Bed velocities of 26.12 m/s at the invert and 22 m/s at the lip were observed in physical model and in numerical model bed velocities of 27 m/s at the invert and 25 m/s at the lip were observed for the design discharge of 8750 m3/s under ungated operation of the spillway. Velocity distribution in terms of magnitude was computed in numerical model at different locations. Velocity magnitudes and vectors as obtained from the numerical model for discharge of 8750m^3/sunder ungated operation for Centre line plane view is shown in the figure 10 and 3D view for the same is shown in figure 11. It is found that the range of the simulated velocities matches with the physical for model values.

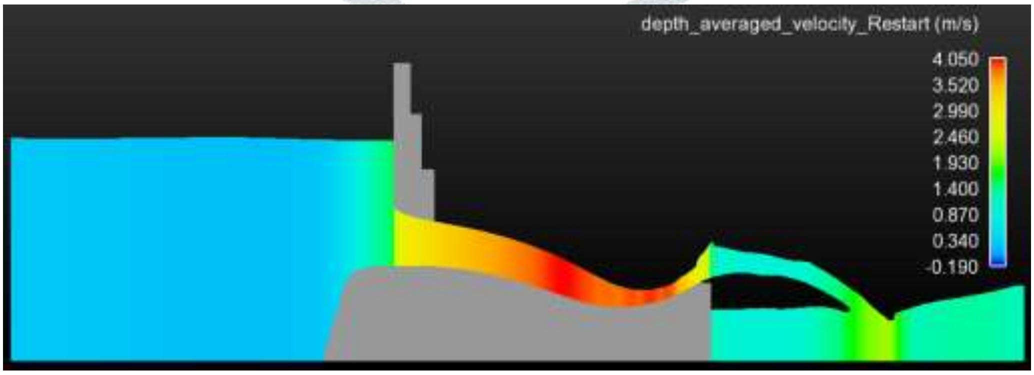

Figure 10: Centre line plane view of velocity distribution from numerical simulation for the discharge 8750m^3/s under ungated operation

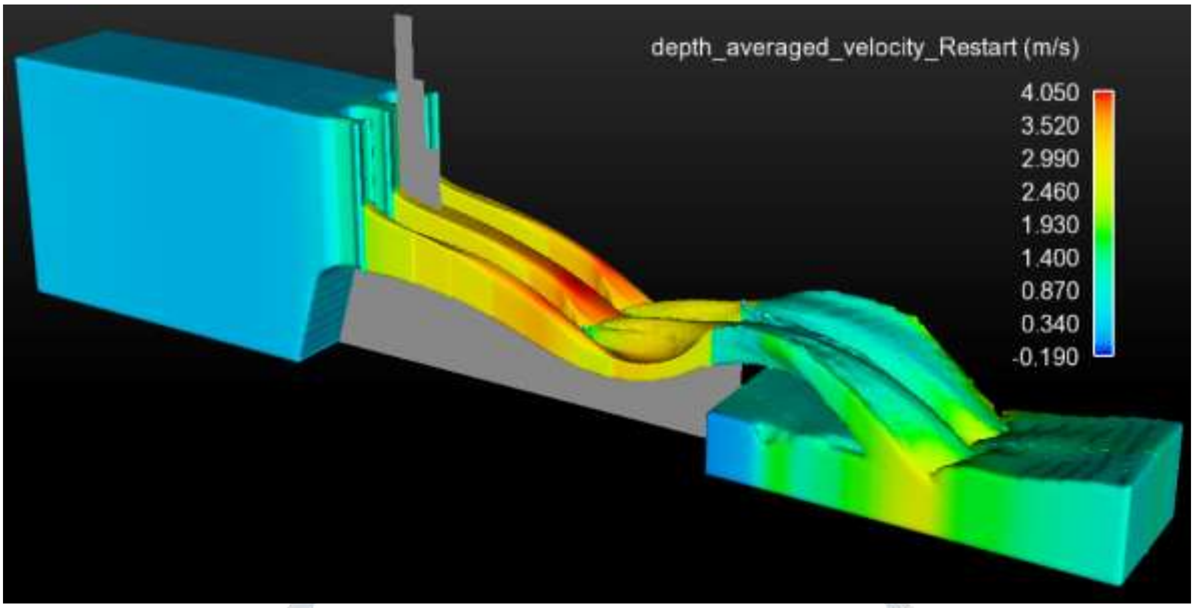

Figure 11: 3D view of velocity distribution from numerical simulation for the discharge 8750 m^3/s under ungated operation

4.5PERFORMANCE OF SPILLWAY AND SKI-JUMP BUCKET

4.5.1 UNGATED OPERATION OF THE SPILLWAY

The performance of spillway and ski-jump bucket was observed for the entire range of discharges and reservoir water levels with ungated operation of spillway. Formation of hydraulic jump in the bucket and cascading of flow over the bucket lip was observed for the discharges up to 3720 m^3/s (RWL El. 1532.4 m) with ungated operation of spillway with gradually increasing discharges. However, due to hysteresis effect, at the time of decreasing discharges by lowering the reservoir water level with ungated operation of spillway, ski-action was seen persisting for the discharges up to 1678 m^3/s (RWL El. 1525.9 m). The reason for unsatisfactory performance of the ski-jump bucket for ungated operation may be attributed to low incoming velocity and large depth of flow.

Ski-action was observed for 50%, 75% and 100% of design discharge i.e. 8750 m^3/s. But fluctuating rooster tails emerging from the end of piers were causing disturbance to the ski-action. The disturbance in the ski-jump jet due to the interruption of rooster tails was increasing with discharge making the top of the water surface rough and wavy there by not allowing for smooth ski-action. It is felt that it is necessary to modify the spillway and the energy dissipator to achieve satisfactory energy dissipation under ungated operations of spillway. Reducing the bucket lip angle form 35° to 30° may be considered as an option to achieve satisfactory energy dissipation. This may facilitate efficient removal of sediment in the bucket during flushing operation.

Trunnion was found to be free for all the discharges. The vertical rise of the ski-jump jet was contained well below the training wall elevation for the discharges of 25%, 50%, 75% and for 100% of design discharge, it was overtopping at the end of the training wall. Intermittent mild vortex formation was observed in the vicinity of right end span at Ch 20-30m upstream of dam axis for discharges of 50%, 75% and 100% of design discharge. It was observed that water profile follows the breast wall bottom profile for 75% and 100% of the design discharge. Hence the breast wall bottom profile is found to be acceptable There was no influence of tail water on the performance of ski-jump bucket as the water levels in the bucket were higher than the tail water levels for ungated operation of spillway.

Numerical simulations are carried out to observe two distinct phases of air and water and flow conditions over the spillway and ski-jump bucket. Phase diagram is obtained for 100% un-gated operation of spillway. Figures 12 and 13 show flow conditions over spillway and energy dissipater in the physical model and the phase diagrams for a discharge of 8750 m^3/s (100%) for ungated operation of spillway from the numerical model respectively.

Figure 12: Photo showing flow conditions over spillway and energy dissipator for a discharge of 8750 m^3/s for ungated operation of spillway

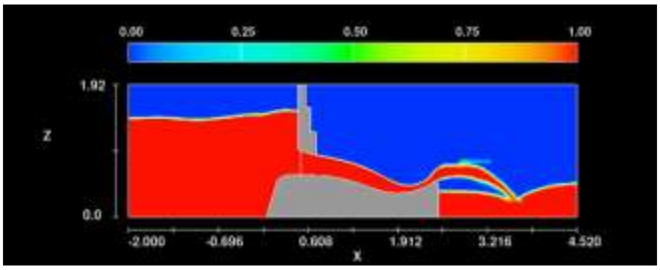

Figure 13: Phase diagram showing flow conditions over spillway and energy dissipator for a discharge of 8750 m^3/s for ungated operation of spillway

4.5.2 GATED OPERATION OF THE SPILLWAY

The performance of ski-jump bucket was observed for the entire range of discharges with reservoir water level at FRL El. 1572 m with gated operation of spillway. Ski-action was observed for entire range of discharges. However, performance of ski-jump bucket was hampered due to the formation of rooster tails for all the discharges. The observed throw distances are more than that in the case of ungated operation of spillway as the head maintained is more on the upstream with FRL El 1572 m. The ski-jump jet was splashing / overtopping over the right training wall at the end, for 75% and 100% of design discharge due to interaction of the fluctuating rooster tails with the main flow. In view of this, it is felt that keeping the pier length same, the pier shape may be modified by extending the tapering to the end of the pier nose in place of the present curved shape, which may eliminate the formation of rooster tails. This modification needs to be studied on the model to assess its performance.

5. RESULTS ANDDISCUSSIONS

Performance of the spillway was assessed in terms of discharging capacity and pressures over the spillway ,water surface profiles, velocities, and flow conditions from physical model studies and also computed from numerical model studies for Q = 8750 m^3/s for ungated operation of spillway. Results of the both the model studies are compared. The following observations are made.

- Discharge observed in physical model is 8750 m^3/s (100%) at RWL 1552.33 m for ungated operation of spillway while in numerical model a discharge of 9154 m^3/s has been realized for the same RWL. So, 4.6% error was observed in comparison of physical and mathematical model discharges. As this deviation is small and hence the discharging capacity of the spillway can be considered adequate.

- It is found that the results obtained from the physical and numerical model for water surface profile obtained under ungated operation of spillway for the discharge of 8750 m3/s (100%) are in close agreement throughout the length of the spillway, except at few chainages. The reasons for the error generation in the numerical model studies are primarily caused by the grid density, especially in the regions close to the interface. The errors can be reduced with a denser grid, but the computing time will be increased accordingly. Rooster tail formation in the physical model also could be reason for the deviation in the water profiles.

- And results obtained from numerical model and physical model are in reasonably good agreement with one another on the spillway profile.

- Pressure profiles obtained by physical as well as numerical modeling are in good agreement with each other and no negative pressure was observed on entire spillway profile for the discharge 8750 m^3/s ungated condition.

- Velocities obtained by physical as well as numerical modeling are matched with each other satisfactorily for ungated operation of spillway for the discharge of 8750 m^3/s (100%)

- Phase diagrams show that flow conditions/features observed on physical model have been captured exactly in the numerical simulation.

- From the velocity distribution figures, it is found that the range of the simulated velocities matches well with the physical model values.

- Formation of hydraulic jump in the bucket and cascading of flow over the bucket lip was observed for the discharges up to 3720 m^3/s (RWL El. 1532.4 m) with ungated operation of spillway with gradually increasing discharges. However, due to hysteresis effect, while lowering the reservoir, ski-action was seen persisting for the discharges up to 1678 m^3/s (RWL El. 1525.9 m). Ski-action was observed for 50%, 75% and 100% of design discharge i.e. 8750 m^3/s. But fluctuating rooster tails emerging from the end of piers were causing disturbance to the ski-action. In view of this, it is felt that it is necessary to modify the design of spillway and the energy dissipator to achieve satisfactory energy dissipation under ungated operations of spillway. Reducing the bucket lip angle from 35° to 30° is be considered as an option to achieve satisfactory energy dissipation under ungated operation of spillway. This may facilitate easy removal of sediment in the bucket during flushing operation.

- Ski-action was observed for entire range of discharges with gated operation of spillway. However performance of ski-jump bucket was hampered due to the formation of rooster tails for all the discharges.

- The ski-jump jet / rooster tail was splashing / overtopping over the right training wall at the end, for 100% of design discharge due to interaction of the fluctuating rooster tails with the main flow for both gated and ungated operation of the spillway and for 75% of the design discharge with gated operation of the spillway. In view of this, it is felt that keeping the pier length same, the pier shape may be modified by extending the tapering to the end of the pier nose in place of the present curved shape, which may eliminate the formation of rooster tails. This may also improve the ski-action thereby improving the performance of the bucket. This modification needs to be studied on the model to assess its performance.

- In view of the above, it is necessary to optimize the design of spillway and the energy dissipator by modifying the present design so as to improve the performance of spillway and energy dissipator. Available options for modifications of spillway and energy dissipator are summarized as below :

  I. Modifying the spillway profile by making it flatter in the susceptible zone of cavitation as mentioned above to prevent the spillway surface from cavitation damage.

  II. Reducing the bucket lip angle from 35° to 30° is an option to achieve satisfactory energy dissipation under ungated operation of the spillway. This may facilitate the efficient removal of sediment in the bucket during the flushing operation.

  III. Keeping the pier length same, the pier shape may be modified by extending the tapering to the end of the pier nose in place of the present curved shape, which may eliminate the formation of rooster tails. This may also improve the ski-action thereby improving the performance of the bucket. This modification needs to be studied on the model to assess its performance.

6. CONCLUSIONS

In the present study, performance of a spillway and energy dissipator of a Run-off-the river hydroelectric project, Bhutan, was evaluated by physical and numerical model studies. Results of the both the model studies are compared. It is found that Numerical / CFD modelling software Flow-3D could effectively simulate the flow conditions over the spillway. Comparison of the results from the two modelling approaches has provided a very good insight into the design process to optimize the spillway design. This parallel use of physical and numerical modelling techniques has provided invaluable insight and greater confidence for future use of standalone CFD analysis in spillway design as a complementary tool for physical model studies. After studying the performance of the orifice spillway of the Run-off-the river Hydro-electric Project, both through physical and numerical techniques, it is found that numerical/CFD modelling can generally be used as a complementary tool to test the preliminary design and in the optimization phase of the spillway design prior to taking up of physical model studies.

ACKNOWLEDGEMENTS

The authors are thankful to Shri B. S. Sundarlal, Scientist 'B', SED division, CW&PRS for constant encouragement and guidance during the course of studies as well as in the preparation of the Technical paper and Director, Central Water and Power Research Station, Khadakwasla, Pune-24 for permitting me to undertake this project work at CWPRS and providing all the facilities preparion of the Technical paper.

REFERENCES

Bhosekar V.V. (2015), "Numerical Modeling of flows on spillways", CWPRS Memorandum "Spillways and Energy Dissipators".

Bhajantri M. R. (2015), "Hydraulic Design Considerations and Modelling Techniques for Spillways and Energy Dissipators", CWPRS Memorandum "Spillways and Energy Dissipators".

Bhajantri M. R. (2015), "Hydraulic Design Aspects of Orifice Spillways for Run-of-river Projects", CWPRS Memorandum "Spillways and Energy Dissipators".

Kulhare Amit et.al.(December 2014) "Experimental and Three Dimensional Numerical Studies for a Sluice Spillway" International Journal of Engineering Research (IJER), Issue-Special 3.

Kulhare Amit et.al.(December 2014) "Numerical Flow Simulation through a Breastwall Spillway – An Application of CFD Software Flow - 3-D" 19th HYDRO 2014 International, MANIT Bhopal

Technical Report No. 5902, arch 2021, Hydraulic Model Studies for Kholongchhu Dam Spillway, Bhutan, 1:45 Scale 2-D Sectional Model, CW&PRS Pune.