3-D Spillway Simulations of Ratle HEP (J&K) for the Assessment of Design Alternatives to be Tested in Model Studies

Amar Pal Singh [1], Faisal Bhat [2], Ekta Gupta [3]

[1] Head – Hydro Engineering
Larsen & Toubro Infrastructure (Hydel BU)
[2] Senior Design Engineer - Civil
Larsen & Toubro Infrastructure (Hydel BU)
[3] Design Engineer - Mechanical
Larsen & Toubro Infrastructure (Hydel BU)

Abstract

Model study of spillways has been successfully carried out in India for the past fifty years at various research institutions. Numerical simulation on the other hand is still used as a complementary tool for hydraulic design and performance investigation. In spite of the advantage of high performance computers that can minimize effort, cost, and time, its accuracy is vulnerable to improper selection of solver, solution parameters and boundary conditions, which may amplify numerical approximations inherently involved in finite volume/finite difference methods. This paper presents a case study of iterative simulations of the flow over spillway of Ratle Hydro Electric Project (850 MW) - (J&K), for various suitable modifications which were intended to obtain a hydraulic design, compatible with the topography on the downstream side. Model studies on a comprehensive model (geometrical scale 1:55) were performed at the Irrigation Research Institute, Roorkee, which revealed that the trajectory formed by the upper level spillway and one of the five main orifice spillways strikes the left bank of the river. To avoid slope stability issues, modifications to the divide wall, left abutment and lip of the upper spillway were first attempted in successive numerical simulations performed in the CFD package Flow-3D Version 10.1, before being proposed to be tested on physical model, thereby substantially reducing time & cost.

Keywords: *Spillway Simulation, Numerical Model, Model Studies, Comprehensive Model, Flow-3D, Spillway Trajectory*

INTRODUCTION

The waterway of the dam in Ratle HEP consists of five orifice spillway bays and one upper level Ogee overflow spillway. It was revealed in the early physical model studies that the trajectory formed by water coming from (a) bay 4 of orifice spillway only and (b) upper level spillway only, strikes the left bank of the river. This could lead to slope failure of the left bank from time to time, in the plunge pool area and thereby add up to the maintenance cost during the operational period.

In-order to avoid such problems in future, modifications in the hydraulic design of spillway were attempted to divert the trajectory of water sufficiently away from the left bank. This was done by gradually increasing the width of (a) divide wall between spillway bay 5 & upper spillway and, (b) left abutment wall, up-to the end of bucket to guide the nappe away from the left bank. As a design alteration, increasing the width of the abutment wall alone would have increased the risk of cavitation damage due to the resulting high discharge intensity. Therefore, a rotation in the lip of the bucket of the upper spillway was also attempted in iterative simulations for various combinations of angle of rotation and the gradually increased width of the divide wall and left abutment, near the lip.

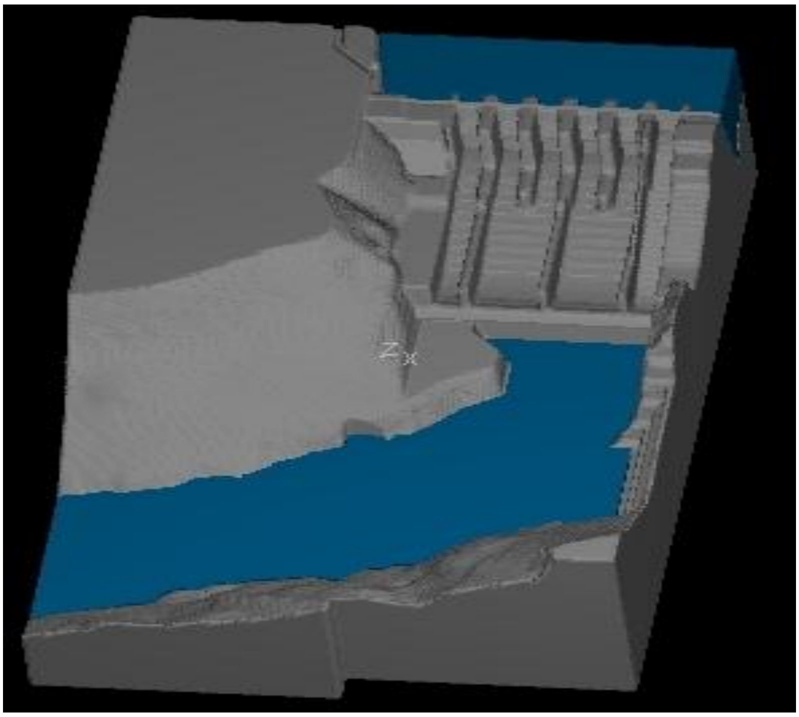


FIG.1 A view of the 3D model that was used in the numercial simulations.

GEOMETRY

The 3D geometry of the reservoir (see figure 1.) was imported as (.topo) file from AutoCAD Civil-3D software and converted to (.stl) format with the newly included geometry tool of Flow-3D v10.1. The 3D geometry of dam was directly imported as (.stl) file from AutoCAD 3D.

MESHING

In-order to verify the numerical model, various conditions involving all the spillway bays were simulated on a coarse mesh having 975,000 active cells (uniform rectangular mesh of size 2m), to avoid prohibitive run-times that would have resulted from a fine mesh covering all the bays. The data so obtained was compared with that obtained from the physical model study. Once the model got verified, the subsequent simulations involving only the upper spillway and the orifice spillway bay no. 4 & 5, were performed using a fine mesh (uniform mesh of size 1m), covering only the desired regions in its domain.

ROUGHNESS

An equivalent uniform roughness of 60 cm for the reservoir has been computed from Manning's 'n' as 0.04 for banks and 0.035 for the bed[1] *and an estimated hydraulic radius, using the following equation:*

$$ROUGH \approx 3.72067 \cdot D_H e^{\left(\frac{-0.103252 D_H^{1/6}}{n_{manning}}\right)} \text{ (fully rough turb., SI units)}$$

Detailed description of the above equation may be found in the Flow-3D User's Manual[2].

The above equation yielded a very low value of roughness for the concrete surface of spillway. Considering the fact that the objective of the simulation was least sensitive to the roughness parameter, a value of 1mm for concrete surfaces and 60 cm for the reservoir was chosen as a reasonable estimate.

TURBULENCE

From the various time averaged RANS (Reynold's Averaged Navier-Stokes equations) turbulence models available in Flow-3D, the most robust and widely applicable turbulence model is the RNG (Renormalized Group) model. The model requires a maximum turbulent mixing length, which represents the upper limit of the size of an expected turbulent eddy in the simulation and is used to limit the amount of turbulent energy created by the turbulence models. As it is recommended that this value should be 7% of the hydraulic diameter (M. H. ShojaeeFard and F. A. Boyaghchi, 2007), an average value of 0.3m was calculated and adopted for all the simulations.

SOLVER OPTIONS

The generalized minimal residual (GMRES) pressure solver for continuity equations, available in Flow-3D is computationally much more efficient than the successive over-relaxation (SOR) algorithm for large domains, because it is able to converge with far less iterations. Hence, for all the simulations ,GMRES solver was used along with automatic limited compressibility to improve the convergence of the overall model. For simulations involving sharp free surfaces, the best implicit advection option is the one with limited advection at free surfaces controlling the time-step size for accuracy (Flow Science Technical Note # 74). The same was adopted in all the simulations, and for the calculation of viscous stresses the default explicit solver was used to maintain accuracy.

BOUNDARY AND INITIAL CONDITIONS

Setting the appropriate boundary condition can have a major impact upon the extent to which the simulation results reflect the actual situation, one is trying to simulate. The upstream boundary at 55m from the dam-axis has been specified as a pressure (stagnation) boundary (see figure 2) with a specified water level corresponding to the case of model testing that is being simulated. In the physical model, water levels for different values of discharge were recorded at this location, although the reservoir extended up to 1000m upstream of dam-axis.

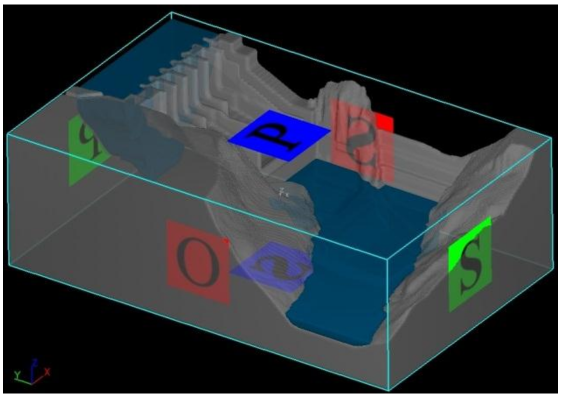


FIG.2 Boundary conditions.

Thus a major approximation which could affect the discharging rates was introduced due to prohibitive computational effort that would have been required to model the reservoir conditions similar to the physical model. The downstream boundary condition was specified as an outlet boundary. All the other boundaries of the domain, except the top boundary which has been set at void pressure (i.e. atmospheric pressure) have been set to symmetry boundary condition, which minimizes computational effort as compared to a wall boundary condition The domain on the downstream of dam has been initially kept filled with water up to an elevation of 940m a.m.s.l to *accelerate steady-state of flow. While on the upstream, the reservoir is filled in each simulation up to the water level recorded in the physical model corresponding to different discharges.*

VALIDATION OF NUMERICAL MODEL

The numerical model was validated by simulating different conditions of spillway operation that were also run in the physical model, and then comparing the discharges, velocities, the maximum level and the maximum length of the nappe of trajectory with the data obtained from model studies(Table 1 to Table 4 shows the comparison between the results of the numerical simulation and the model studies).*A comparison between the discharge value obtained from model studies, numerical simulation and empirical equations as per Indian Standard IS 6934[6] is also given in Table 5. All the simulations were run for a minimum run-time of 500 seconds to ensure that the flow has reached a steady-state. This was also confirmed by observing steady values of mean kinetic energy and volume of fluid in the domain. Minor errors were anticipated in the discharge values which could be attributed to the coarse size of the mesh and the approximation of the upstream boundary condition. The boiling action of water in the plunge pool was quite well simulated with water levels rising up to a maximum of El. 969m a.m.s.l (see figure 3), compared to 970m in model studies. More importantly the trajectory of water hitting the left bank of river was very well simulated by the numerical model, and could easily be verified by a comparison with the photographs available from model testing.*

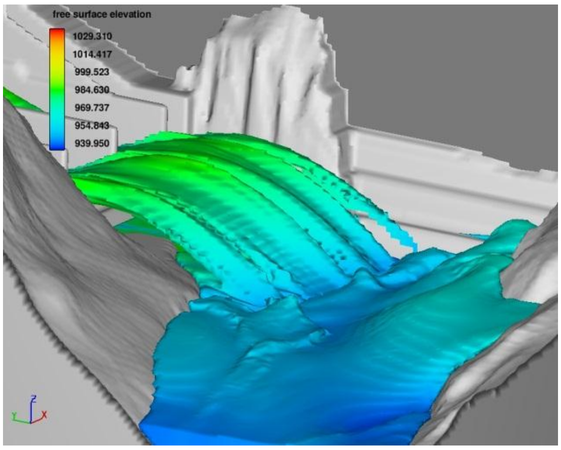
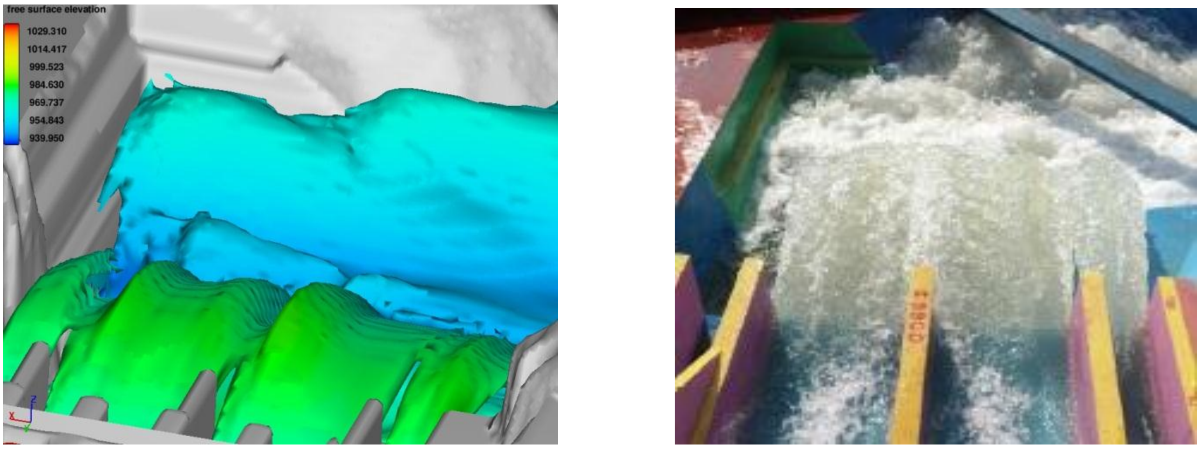


FIG.3 A view of the boiling action taking place in the plunge-pool.

The absence of noise visible in the trajectory in the model studies could be ignored as it could have only been simulated on a mesh of size as small as 0.1m. Thus the numerical model was considered sufficiently accurate for the purpose of recording water trajectories corresponding to different modifications done to the spillway design.



(a) (b)

FIG.4 Trajectory of water hitting the left bank in (a) numerical simulation and in (b) model studies.

DESIGN MODIFICATIONS

As a measure to divert the trajectory away from the left bank, initial simulations involved a gradual increase in the width of (a) the divide wall between spillway bay no. 5 and upper spillway and (b) the left abutment, from the point where the width of piers chamfers in the original design.

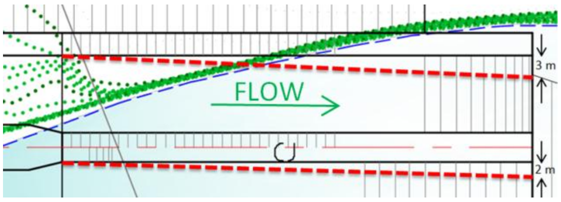


FIG. 5 The width of the left abutment and the left-most divide wall was increased by 3m and 2m, respectively.

Since it was also desired to limit the resulting increase in discharge intensities, the lip of the upper spillway was rotated along an axis perpendicular to the dam-axis and passing through the right corner of the lip, as an additional design alteration.

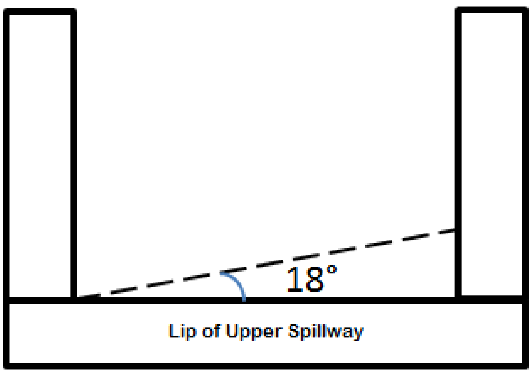


FIG. 6 A super-elevation of 18 degrees was provided at the lip of the upper spillway.

A number of simulations were run with different combinations of rotation angle of the lip and widths of the divide wall and the left abutment. A rotation of 18 degrees along with an increase of 3m and

2m in the width of left abutment and divide wall respectively, were found to be the minimum design alterations to sufficiently deflect the trajectory away from the left bank of the river.

Table 1. Comparison of discharging capacity of the spillway under different conditions of operation.

S. No	Condition of Gate Opening	Pond level at 55 m U/S of Dam (m)	Discharge (cumec) Numerical Model	Discharge (cumec) Physical Model	Error (%)
1	All six bays fully open	1029.0 (FRL)	18115.0	16550.0	9.5
2	All main orifice spillway fully open, upper level spillway closed only	1024.5 (FRL)	14510.0	15080.0	-3.7
		1015.86 (MDDL)	12770.0	11750.0	8.6
3	Upper level spillway open only	1029.0 (FRL)	1348.0	1425.0	-5.4
4	One right bay of main orifice closed only	1029.0 (FRL)	14680.0	13650.0	7.5

Table 2. Comparison of the maximum level of upper nappe of trajectory.

S. No	Condition of Gate Opening	Pond level at 55 m U/S of Dam (m)	Max. level of upper nappe of trajectory (m) Numerical Model	Max. level of upper nappe of trajectory (m) Physical Model
1	All six bays fully open	1029.0 (FRL)	985.0	985.6
2	All main orifice spillway fully open, upper level spillway closed	1024.5 (FRL)	984.0	985.4
		1015.86 (MDDL)	982.5	984.5
3	Upper level spillway open only	1029.0 (FRL)	981.0	982.6
4	One right bay of main orifice closed only	1029.0 (FRL)	985.5	986.0

Table 3. Comparison of the maximum length of water trajectory from the dam axis.

SL. No	Condition of Gate Opening	Pond level at 55 m U/S of Dam (m)	Max. length of trajectory from dam axis (m) Numerical Model	Max. length of trajectory from dam axis (m) Physical Model
1	All six bays fully open	1029.0 (FRL)	232.0	235.0
2	All main orifice spillway fully open, upper level spillway closed only	1024.5 (FRL)	226.0	230.0
		1015.86 (MDDL)	216.0	210.0
3	Upper level spillway open only	1029.0 (FRL)	238.0	236.0
4	One right bay of main orifice closed only	1029.0 (FRL)	236.0	225.0

Table 4. Comparison of Discharge through each bay of the Main Spillway with u/s pond level @ EL. 1024.5m

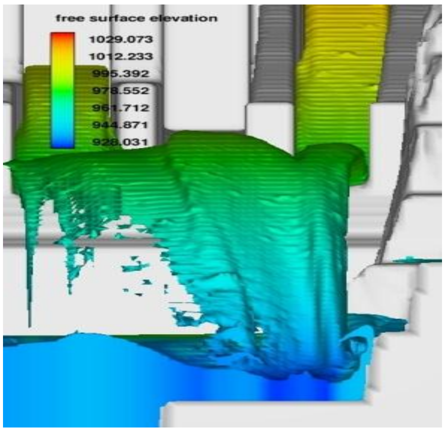
Bay no. Right to Left	Velocity (m/s) **Numerical Model**	Velocity (m/s) **Physical Model**	Discharge (cumec) **Numerical Model**	Discharge (cumec) **Physical Model**	Discharge Error (%)
1	**18.74**	**17.91**	**3045.0**	**2734.0**	**11.4**
2	**19.80**	**17.91**	**3080.0**	**2734.0**	**12.6**
3	**21.43**	**19.33**	**3090.0**	**2950.0**	**4.7**
4	**21.30**	**19.01**	**3003.0**	**2902.0**	**3.5**
5	**22.90**	**20.90**	**3125.0**	**3190.0**	**-2**

Table 5. Comparison of the Discharging Capacity of Upper Level Spillway (u/s pond level @ El. 1029m)

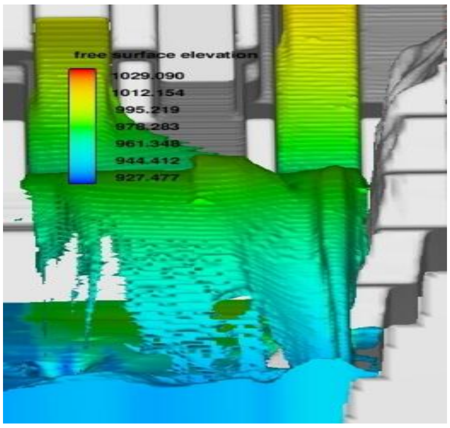
Gate Condition	Discharge (cumec) **Theoretical**	Discharge (cumec) **Numerical Model**	Error w.r.t Theoretical Value (%)	Discharge (cumec) **Physical Model**	Error w.r.t Theoretical Value (%)
Upper level spillway open only	**1237**	**1348**	**8.97**	**1425**	**15.19**

CONCLUSION

The results obtained from the numerical simulations gave enough confidence to the engineers to proceed further with model testing of the spillway with the final modifications evolved from iterative simulations. Although, the use of a coarse mesh caused errors in the discharge values, the trajectory of water was simulated quite accurately by Flow-3D, (see figure 4 to compare results of model testing and numerical simulation). Flow-3D was therefore used successfully to aid the model testing of spillway for design alternatives. Figure 7 (a &b) shows how the trajectory of water coming out from the spillway, which hit the left bank in the plunge-pool area, got deflected away from the bank by making suitable design alterations.



(a)



(b)

FIG.7 Front view of water trajectory (a) with modified divide-wall, left abutment and spillway lip and (b) without any design modifications.

References

1. Chapter X “Simulation Studies and Techniques” of CBI&P Publication No. 204, River Behavior, Meandering and Management, 1989.
2. Chapter 12 “Troubleshooting” ofFlow-3D User’s Manual version 10.0,Physical Modeling Questions : “How do I select a surface roughness that correlates with Manning’s n”?
3. Hydraulics Tutorial “Energy Dissipation in a Drop-Structure Flow” of Flow-3D User’s Manual version 10.0.
4. Shojaeefard M.H. and Boyaghchi, "Studies on the Influence of Various Blade Outlet Angles in a Centrifugal
Pump when Handling Viscous Fluids", American Journal of Applied Sciences, 2007, ISSN 1546-9239.
5. Flow Science Technical Note # 74 “Implicit Advection in Flow-3D: Its Uses and Limitations”, 5/17/06 CWH.
6. IS 6934 : 1998 “Hydraulic Design of High Ogee Overflow Spillways – Recommendations”