CFD ANALYSIS FOR HYDRAULIC DESIGN OF SURGE TANK FOR HYDROELECTRIC PROJECTS

Shital S. Nadagire[1], Ashish A. Doshi[2], Pramod B. Deolalikar[3], Dr. Karan Gupta[4]

[1] Infraplan Hydraulic Laboratory, Pune, India
[2] Infraplan Hydraulic Laboratory, Pune, India
[3] Senior Independent Hydraulic Consultant, Pune, India
[4] STVENANT LLP, Delhi, India
shitalnadagire@infraplan.in

Abstract. The hydraulic design of a surge tank requires accurate assessment of transient flow conditions to ensure reliable turbine operation under varying reservoir levels, particularly during 20% overloading and extreme conditions like instantaneous load rejection or acceptance. Key design parameters include maximum and minimum water levels (MWL & MDDL), head losses, and the discharge coefficient (Cd) of restricted orifices. Traditional 1D modelling assumes constant Cd, which oversimplifies real flow behavior. In contrast, Computational Fluid Dynamics (CFD) enables detailed analysis by simulating transient, pressure-driven flow through the surge tank and orifice, accounting for factors like geometry, Reynolds number, and flow direction. Using turbulence models such as k-ε or k-ω SST, dynamic mesh techniques, and pressure-based flow resolution, CFD calculates time-dependent Cd values based on instantaneous pressure and velocity distributions rather than fixed assumptions. This study focuses on a hydroelectric system with a restricted orifice surge tank upstream of three penstocks, covering the entire 7700 m headrace tunnel from the desilting basin to the main inlet valve. CFD simulations modelled load rejection/acceptance using turbulence models, dynamic meshing, and pressure-based flow solvers. These simulations provided insights into time-dependent Cd variations and associated head losses during upsurge and downsurge events. Parametric studies on various orifice geometries helped evaluate flow behavior, validating findings such as those by Diao (2016) regarding orifice-wall spacing and flow coefficient stability. Ultimately, CFD enabled the optimization of orifice profiles and surge tank geometry, highlighting its value in achieving accurate, reliable surge tank designs in hydropower systems.

Keywords: Computational Fluid Dynamics (CFD), Fluid structure interactions, Transient scenarios, Orifice profile, Discharge coefficient (Cd).

1 Introduction

Hydroelectric projects are vital for sustainable energy, with safety and efficiency largely dependent on the water conductor system Head Race Tunnel (HRT), surge tank, and pressure shafts. The surge tank plays a key role in mitigating water hammer effects from sudden turbine load changes. Hydraulic transients, such as pressure surges and depressions, can cause severe stress, risking damage or failure. Accurate prediction and mitigation of these phenomena are therefore critical. Traditional one-dimensional methods like the method of characteristics provide preliminary insights but oversimplify complex three-dimensional flow behaviors, leading to discrepancies. Computational Fluid Dynamics (CFD), by solving Navier-Stokes equations, offers detailed simulations of flow dynamics, pressure distributions, and velocity profiles. Its application enhances turbine efficiency, identifies localized stresses, and optimizes design, reducing environmental impact. This paper examines the impact of CFD in hydropower design, using the Shongtong Karcham Hydro Electric Project (450 MW) in Himachal Pradesh, India, as a case study. This paper presents the CFD-based hydraulic study of the water conductor system, with particular focus on surge tank performance, orifice profiling, and discharge coefficient estimation under various transient scenarios. The study was conducted by Infraplan Hydraulic Laboratory, Pune, in association with HPD Consultants Pvt. Ltd., Pune.

2 Literature Review

2.1 General Principles of CFD in Hydropower Engineering

Computational Fluid Dynamics (CFD) has become a transformative tool in hydropower engineering, enabling detailed simulation of fluid flow by numerically solving the Navier–Stokes, continuity, and energy equations. Unlike traditional analytical or physical model studies, CFD provides precise insights into flow patterns, pressure distributions, velocity profiles, and turbulence within complex geometries. Its applications in hydropower are wide-ranging, from optimizing turbine components such as runners, guide vanes, and draft tubes to predicting sediment transport, cavitation, and erosion, all of which are critical for efficiency and long-term reliability. By visualizing three-dimensional flow fields, CFD empowers engineers to make informed design decisions that enhance performance and robustness of hydropower plants.

Equally important is transient analysis, particularly in addressing water hammer phenomena. Hydropower systems operate under dynamic conditions, with load variations, grid fluctuations, and emergency shutdowns inducing hydraulic transients. Water hammer, caused by abrupt changes in flow due to rapid valve or pump operations, generates pressure waves that can exceed normal operating levels, leading to cavitation, structural damage, or even catastrophic failure. Comprehensive transient analysis is therefore essential to predict and mitigate these risks. While one-dimensional methods such as the method of characteristics provide preliminary system-wide insights, they often oversimplify complex flow behavior. CFD integration offers a more accurate approach, capturing localized pressure fluctuations and intricate three-dimensional interactions, thereby ensuring safer, more efficient, and more reliable hydropower system design and operation.

2.2 Previous Studies and Advancements in CFD Application for Water Conductor Systems

The role of Computational Fluid Dynamics (CFD) in hydropower engineering has advanced from academic validation studies to a core tool in practical design and analysis. Early work focused on comparing CFD results with physical model experiments, proving its accuracy in simplified geometries. With improved computational power and algorithms, CFD is now applied to complex hydropower components, including intake structures, bifurcations, valves, gates, and long headrace tunnels, enabling precise evaluation of head losses, flow distribution, and cavitation risks. A major development has been the integration of CFD with transient analysis. While traditional one-dimensional models capture overall pressure wave propagation, CFD provides detailed insights into localized fluctuations and three-dimensional flow structures. This has proven especially valuable in surge tank design, where CFD helps optimize orifice profiles and discharge coefficients (Cd) for effective surge damping. CFD also complements analytical and experimental methods by refining 1D models and improving prediction accuracy. This combined approach using 1D models for system-wide analysis and CFD for critical components represents the current state-of-the-art in hydropower design. Ongoing advances in CFD software and computational capacity promise even greater precision, enhancing the safety, efficiency, and sustainability of hydropower projects worldwide.

3 Hydraulic design aspects of Orifice of Surge Tank

The hydraulic design of an orifice in a surge tank is critical for ensuring efficient operation and stability of the system. Key aspects of orifice design include its size and geometry, as the diameter and edge profile directly influence flow control and pressure regulation. The discharge coefficient (Cd) must be optimized, considering edge sharpness and surface finish to ensure efficient flow. The orifice should accommodate the maximum expected flow rate while maintaining acceptable pressure drops to avoid inefficiencies, and it must withstand transient conditions such as sudden changes in flow or pressure that can trigger water hammer. Material selection is critical to resist operating pressures, temperatures, corrosion, and erosion. Additionally, the orifice should aid in energy dissipation to reduce surges and stabilize system performance, while its design must also allow for easy inspection, cleaning, and replacement to ensure long-term reliability and maintenance efficiency. The components considered for the CFD study is mentioned below with its specifications

- Feeder Tunnels: Four independent feeder tunnels (5.24 m diameter) connecting desanders to the HRT.
- Head Race Tunnel (HRT): A 10.5 m diameter, 7712.70 m long circular shaped tunnel.
- Surge Shaft: A restricted orifice underground surge shaft of 30.6 m diameter with a 6.5 m diameter orifice, located at the end of the HRT.
- Pressure Shafts: Three underground pressure shaft systems, each 5.0 m in diameter and approximately 192 m long, and trifurcating from the HRT.

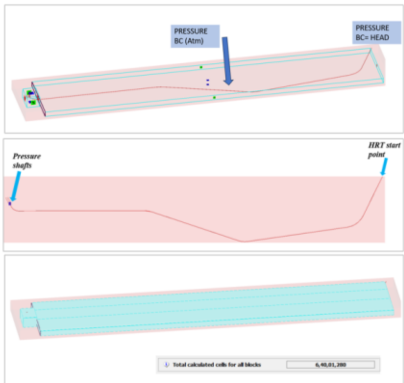
- Power House: Located on the left bank of the Sutlej River, housing three Vertical Francis Turbines, each with a capacity of 150 MW.

4 CFD Model Studies

4.1 Software and Model Configuration with boundary conditions

The CFD analysis was carried out using Flow3D Hydro, a specialized computational fluid dynamics software for hydraulic engineering applications. The complete water conductor system from the desilting basin outlet (upper boundary) to the Main Inlet Valve (MIV) at the turbine entry (downstream boundary) was reproduced in three dimensions. The model encompassed the full 7,712.70 m length of the HRT, the surge tank with restricted orifice, and all three pressure shafts.

The model was discretised into a structured mesh with high-resolution refinement applied at critical zones particularly around the restricted orifice, the junction of the HRT and surge tank, and the pressure shaft bends where velocity gradients and pressure variations are most severe. The total mesh comprised approximately 6,40,01,280 computational cells, ensuring adequate resolution for accurate transient flow capture. The CFD setup employed turbulence modelling with both k-ε and k-ω SST models, suitable for wall-bounded transient flows with strong pressure gradients. A pressure-based solver was used for incompressible flow resolution, while the Volume of Fluid (VOF) method tracked the free surface within the surge shaft.



To capture flow evolution during load transients, dynamic mesh techniques were applied, ensuring accurate representation of moving boundaries. Additionally, a high-resolution computational mesh was generated, with refinement near the orifice to resolve velocity gradients and turbulence structures effectively.

4.2 Orifice Profiling

The CFD studies were carried out for different orifice profiles, to investigate performance of head race tunnel, pressure shaft and surge tank during steady state as well as transient cases. The following transient scenarios were tested and results compared with 1-D analysis using Bentley-HAMMER. During this CFD simulation profile of orifice were used are as follows and shown in below figures.

- Option-1: Represents the baseline sharp-edged rectangular orifice profile as per the original design, with no geometric modification to the edges of the slab.
- Option-2: Introduced curvature on the upper edges of the orifice slab.

- Option-3: Incorporated a 500 mm radius curvature on both the upper and lower edges of the orifice slab

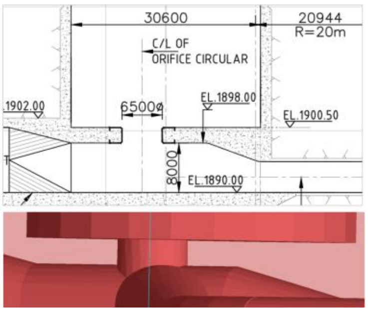


Fig. 1. Orifice profile option 1

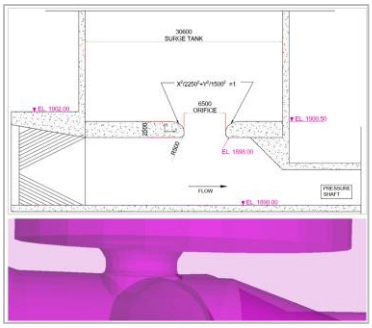


Fig. 2. Orifice profile option 1

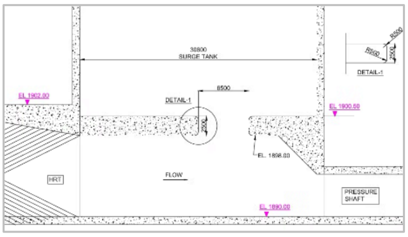
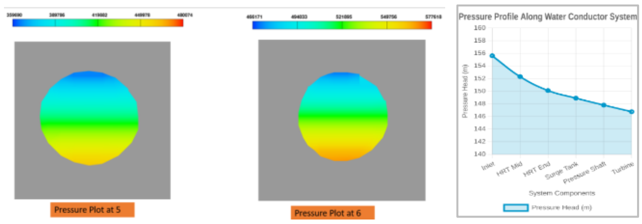


Fig. 3. Orifice profile option 3

5 Result and Discussions

5.1 Steady State Scenario

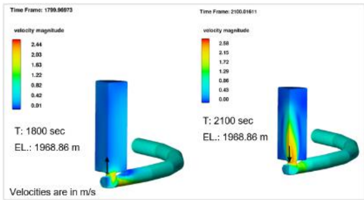


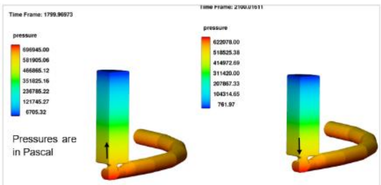
The steady-state simulation was conducted at a design discharge of 393 m³/s with a maintained water surface elevation of 1956.00 m at the Head Race Tunnel inlet, corresponding to the Full Reservoir Level. The model was run until steady flow was achieved, after which pressure profiles, velocity distributions, and head loss data were analyzed. Pressure variations were assessed along three HRT segments, with detailed three-dimensional distributions examined near the surge tank, penstocks, and turbine inlets. Velocity contours in the surge tank and penstock further characterized steady-state flow behavior.

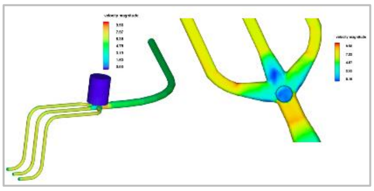
The head loss evaluation showed that the static head available at the turbine inlet, calculated as the difference between EL 1956.00 m and the turbine centreline elevation of EL 1800.40 m, was 155.60 m. The maximum hydraulic pressure recorded at the turbine inlet was 1,438,757 Pa, equivalent to a pressure head of 146.75 m. Thus, the total head loss between the HRT inlet and turbine inlet, including friction and minor losses, was 8.85 m an acceptable value consistent with design expectations. with design expectations.

5.2 Transient Scenarios

The multiple scenarios were conducted for different conditions for considered three orifice profiles. The results from the studies are given below.







For each scenario, the maximum upsurge level, minimum downsurge level, pressure distributions, velocity profiles, and dynamic Cd values were extracted and compared with the prior 1D analyses.

5.3 Estimation of Cd value

One objective of CFD modelling is to verify orifice profiling by estimating the discharge coefficient (Cd) under transient conditions. Cd is determined by extracting pressures at planes upstream and downstream of the orifice, calculating the pressure difference (ΔP) at each time step, and applying the governing equation to derive Cd values. This procedure enables accurate assessment of orifice efficiency and flow behavior during hydraulic transients.

$$Cd = V / (\sqrt{2} \times \Delta p/\rho) \quad (1)$$

Where: V = Velocity, Δp = Pressure difference, ρ = Density

Accordingly, Cd values were calculated for each scenario. The variation of Cd values for load rejection case and load acceptance case is tabulated below in table.

5.4 Comparative Analysis of Results

A key objective of the multiple scenarios was conducted for different orifice profile. Table below represents a comparative summary of hydraulic parameters across all scenarios.

Table 1. Comparative summary of all the scenarios against different orifice profile.

Sr. No.	Scenario Description	Orifice Profile Option	Design Discharge (m^3/s)	Cd – Outflow (Upward)	Cd – Inflow (Downward)	Max. Water Level in surge tank	Min. Water Level in surge tank	Max. Turbine Pressure Head (m)
1	Steady State	Option 1	393	0.70	0.80	-	-	146.75
2	Transient – Load Rejection	Option 1	393	0.806	0.803	1989.90	1934.50	186.52
3	Transient – Load Acceptance	Option 1	393	0.806	0.803	1951.15	1907.41	145.91
4	Transient – Load Rejection (22% Overload)	Option 1	480	0.806	0.803	1995.28	1934.50	192.38
5	Transient – Load Rejection	Option 2	393	0.836	0.832	1991.13	1932.78	186.52
6	Transient – Load Acceptance	Option 2	393	0.836	0.832	1959.78	1907.09	145.91
7	Transient – Load Rejection (22% Overload)	Option 3	480	0.812	0.812	1995.96	1931.74	192.88
8	Transient – Load Acceptance (22% Overload)	Option 3	480	0.812	0.812	1950.38	1894.44	145.91
9	Parametric Study	Option 3	393–480	0.812	0.812	1998.20	—	—

CFD analysis confirms that the surge shaft and orifice arrangement remain effective under both normal and overload transients, keeping upsurge and downsurge within safe limits. Evaluation of three orifice profiles shows that edge curvature slightly improves the discharge coefficient (Cd), which is consistently higher than values assumed in earlier one-dimensional analyses.

6 Conclusion

This paper highlights the effective use of CFD analysis for estimation of various unknown hydraulic parameters such as the coefficient of discharge for a restricted orifice type surge tank and assessment of water levels, and pressures over the entire water conductor system at any desired cross section and location. In the present case, the transient analysis conducted using the CFD demonstrated the efficiency for assessing pressure distributions, velocities, and frictional losses. The studies conducted for the surge tank and orifice arrangement indicated that profiling of the restricted orifice marginally improved the discharge coefficient. The studies for assessing the coefficient of discharge for restricted orifice provided more reliable values than one dimensional analysis. Comparison with Bentley-HAMMER results demonstrated very good agreement. CFD analysis gave very satisfactory results in respect of surge tank geometry and orifice locations and its diameter thickness and modifications such as rounding of edges for all operating conditions of the reservoir and drawl of discharges. Three restricted orifice profiles were studied indicated that introduction of curvatures at the edges improved the coefficient of discharge. The exact assessment removes the doubts of the assumptions generally made in one dimensional analysis. This combined approach represents best practice, reinforcing CFD as an indispensable tool for design optimization, surge mitigation, and accurate prediction of complex hydraulic phenomena in hydropower systems.

Acknowledgments. We gratefully acknowledge the project authorities for referring the studies and for their continuous support and motivation during the model studies. We are also thankful to our advisors, consultants, and project engineers for their continuous support and guidance throughout the study.

References

1. Infraplan Hydraulic Laboratory (2025). Report No. 32/2025: Computational Fluid Dynamics (CFD) Analysis for Water Conductor System of Shongtong Karcham Hydro Electric Project (450 MW), Himachal Pradesh. India. IHL Technical Report, Pune
2. Diao, X.F., Zhang, X.H. and Yang, Y.Y. (2016). Sensitivity Analysis of Restricted Orifice in the Hydropower Station. In: Proceedings of the 5th International Conference on Civil, Architectural and Hydraulic Engineering (ICCAHE 2016), pp. 520–525. Atlantis Press
3. Adam, N.J., De Cesare, G., Schleiss, A.J., Richard, S. and Muench-Alligné, C. (2016). Head loss coefficient through sharp-edged orifices. IOP Conference Series: Earth and Environmental Science, 49(6), p.062009. IOP Publishing
4. Abd, H.M., Alomar, O.R. and Mohamed, I.A. (2019). Effects of varying orifice diameter and Reynolds number on discharge coefficient and wall pressure. Flow Measurement and Instrumentation, 65, 219–226
5. Yang, Z.Y., Cheng, Y.G., Liu, K., Wang, Q. and Yang, F. (2021). Optimization of surge tank structures in hydropower station based on VOF method. IOP Conference Series: Earth and Environmental Science, 627(1), p.012007