International Journal on Engineering Technology and Infrastructure Development (InJET-InDev)
www.lec.edu.np/journal
Volume 2, Issue no. 2, Page 124-137
Received Date: 11/25/2025
Revised Date: 5/11/2026
Accepted Date: 5/26/2026
ISSN: 3021-999X (Print)
https://doi.org/10.3126/injet-indev.v2i2.95708

Hydraulic Performance Evaluation of Approach Flow Using CFD: A Case Study of Budhi Gandaki Hydroelectric Project (341 MW)

Sanish Raj Lakhey[1], Mukesh Raj Kafle[2]

[1] *Department of Civil Engineering, Pulchowk Campus, Institute of Engineering, Tribhuvan University, Nepal, sanishlakhey999@gmail.com,*
[2] *Department of Civil Engineering, Pulchowk Campus, Institute of Engineering, Tribhuvan University, Nepal, mkafle@pcampus.edu.np*

Abstract

The hydraulic performance of settling basins in hydropower projects depend greatly on the approach flow conditions. A three-dimensional RANS-based CFD model was validated with field measurements from the Middle Mewa Hydropower Plant (MMHPP) to study these dynamics. The model showed a good correlation in velocity distribution and turbulence characteristics. The validated model was then utilized for two alternate approach-tunnel configurations within the Budhi Gandaki Hydroelectric Project (BGHEP). The results indicate that culvert geometry significantly influences the flow entering the settling basin. The baseline setup (Case 1) made the velocity distribution uneven, the entrance jets very powerful, and the near-bed recirculation very wide. The optimized setup (Case 2), on the other hand, made the flow more even and lowered the intensity of the turbulence. The research shows that for Himalayan hydropower projects to manage hydraulic performance well; the approach canals must be designed correctly. The results also show how useful validated CFD may be as a design tool for improving headworks hydraulics, especially when testing with physical models is not possible.

Keywords: Approach flow, Settling basin, CFD, RANS turbulence model, Himalayan rivers basin

1. Introduction

Hydropower plants in rivers with a lot of silt, like the Himalayas, sometimes have to deal with significant sediment loads. This wears down hydro-mechanical equipment and makes the plant less efficient overall (Morris & Fan, 1998). Settling basins are an important feature of hydropower headworks. They remove suspended sediment that is floating in the water before it reaches to the headrace system. This makes the turbines last longer and costs less to maintain (Morris, 2016). Many things can affect how well these basins work, including the state of the approach flow, which can have a large effect on how well they work and how much sediment they accumulate. However, the water flow at the basin inflow needs to maintain consistent so that the silt may be removed swiftly. In reality, the water that flows into the settling basin is frequently not uniform since the curves, bends, transitions, and barriers at the intake are all very complicated. These kinds of uneven approach flows can induce short-circuiting, dead zones, and pockets of recirculation inside the basin. These things make settling less effective and let tiny particles flow away (Dulal, 2020). Bad approach flow conditions are one of the most common reasons why many existing basins don't perform as well as they could.

Three-dimensional computational fluid dynamics (CFD) modeling has proven a very useful tool for researching and improving how settling basins behave in the past few years. CFD lets you look at flow patterns, velocity distribution, and particle trajectories in a lot of detail, which is hard to achieve using physical models or field observations. Numerous research has shown that 3D Computational Fluid Dynamics (CFD) may be used to test and improve the designs of settling basins.(Bishwakarma, 2015; Dulal, 2020; Shrestha, 2012). However, limited study has focused on the influence of different intake approach flow conditions on the internal hydraulics and effectiveness of settling basins.

Nøvik et al. (2014) conducted a study that showed how vital it is for settling basins to have flow conditions that change from time to time. The study discovered that the geometry of the inlet might make the formation zones and high turbulence happen. This could make the trap less efficient and maybe even wear down the turbine. The study emphasized the significance of employing computational fluid dynamics (CFD) models to analyze turbulence behavior and velocity distribution. This can help make estimations of how well traps work more precise.

The study aims to assess the performance of a settling basin under various intake approach flow conditions through 3D numerical modeling. The study seeks to enhance the design and functionality of settling basins in sediment-laden rivers by carefully investigating the impact of approach flow on velocity uniformity and turbulence characteristics.

2. Study Area

Budhi Gandaki Hydroelectric Project (BGHEP) is a Peaking Run-of-River plant of installed capacity 341 MW, being developed by Times Energy Pvt. Ltd (TEPL). The project is located about 170km west of Kathmandu, in Gorkha District of Gandaki Zone in Province 4 of Nepal. The headworks site is situated at Jagat village of Sirdibas VDC and Powerhouse is situated at Tatopani village of Uiya VDC in Gorkha district. Project boundary extends between 28° 17′ 00" to 28° 22' 00" north of latitude and 84° 52' 40" to 84° 55' 15" east of longitude. Project is envisaged to be as right bank scheme with underground water conveyance system and powerhouse complex. The project was envisaged to generate about 254 MW installed capacity at the time of desk study report however the installed capacity has been increased to 341 MW after optimization of the project.

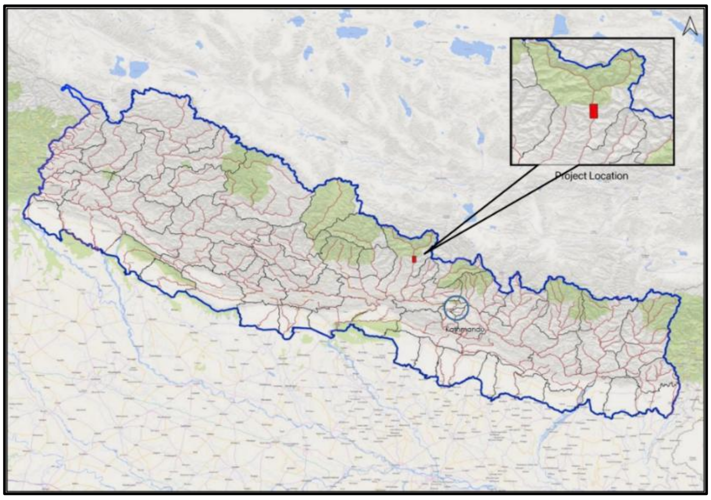

Figure 1. Project Location Map

3. Methodology

This research employs a methodology based on computational fluid dynamics across three spatial dimensions. The model is designed to replicate the flow in the settling basin, as well as the approach flow from the intake of the MMHPP project. For these approach primary drawings has been taken from feasibility study of MMHPP and Physical model test report from Hydro Lab. The software utilized for the investigation is the commercial application ANSYS Fluent. This chapter provides details regarding the examination of the physical model, its development in the CFD, and after the validation of numerical model with physical model later on applied to two cases of BGHEP. The initial step involves configuring the model in ANSYS Fluent by establishing the boundary conditions, creating the mesh, and specifying the input velocity conditions. A simulation is conducted to evaluate whether the selected grid size is optimal by assessing the outlet flow rate. If not, the mesh is established, and the procedure recommences until the appropriate grid size is identified. The numerical model operates on the optimal mesh, enabling visualization of flow characteristics through velocity vectors. Subsequently, the findings are juxtaposed with data from a study utilizing a physical model that employed dye tests to analyze flow dynamics and an Acoustic Doppler Velocimeter (ADV) to measure velocity. If discrepancies arise, the boundary conditions are modified, and the simulations are executed again until the flow characteristics fall within the permissible range. Upon validation, the final data is obtained, displayed, and analyzed to comprehend the fluid dynamics of the system. The overall methodology is illustrated in the Figure 2.

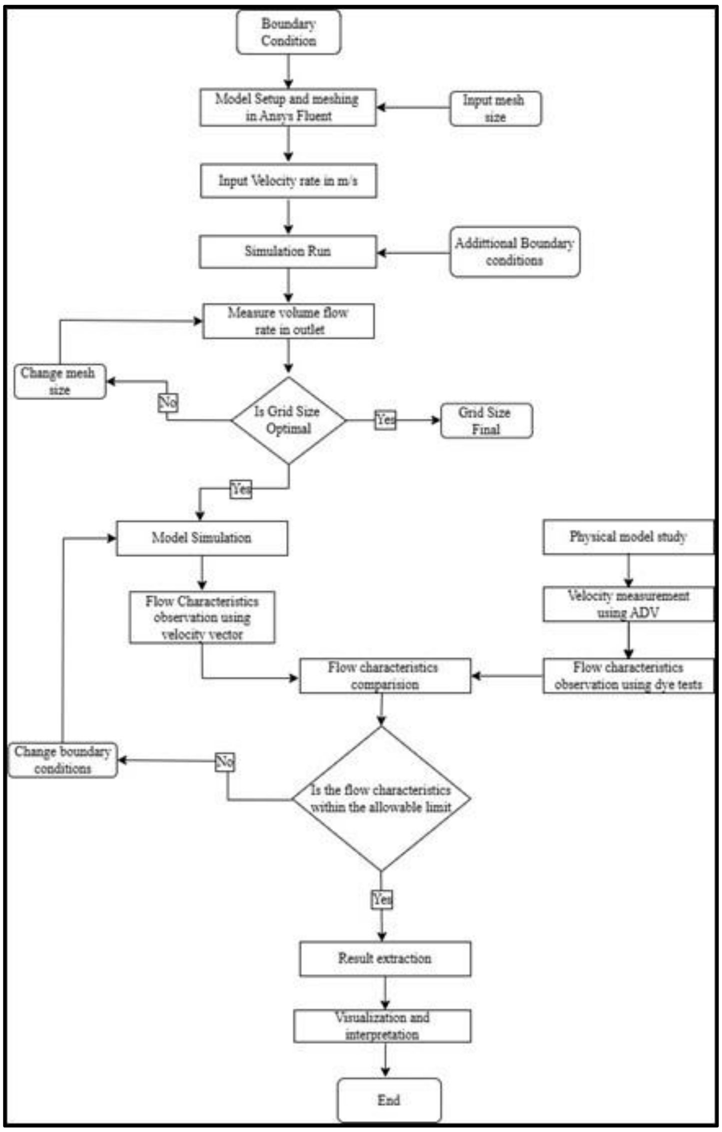

Figure 2. Modelling Framework

3.1 Velocity Measurements on the Physical Hydraulic Model of MMHPP

A 1:35 physical scale model of the settling basins for the planned 48 MW Middle Mewa Hydropower Project (MMHPP) has been constructed at Hydro Lab Pvt. Ltd, Kathmandu, Nepal as shown in Figure 3**Error! Reference source not found.**. There are four bays over two independent settling basins, designed for a total design discharge of 12.5 m^3/s. The velocity measurement from the ADV was conducted at 12.5 m^3/s. The Froude scaling law indicates that the discharge in the physical model is 0.000138 m^3/s.

Figure 3. Arrangement of Physical Model of MMHPP (Scale 1:35)

The data files obtained from the laboratory measurements are subsequently exported utilizing the Horizon ADV software created by Sontek. The output files are formatted in a tabular structure, and subsequent analysis of the results is conducted using Microsoft Excel. Then, the two – Dimensional plan and section of Middle Mewa Hydropower Project intake to settling basin structure is finalized using AutoCAD and 3D modelling was done in Rhino 3D modelling tool. The geometry developed in .3dm extension in imported to the CFD program.

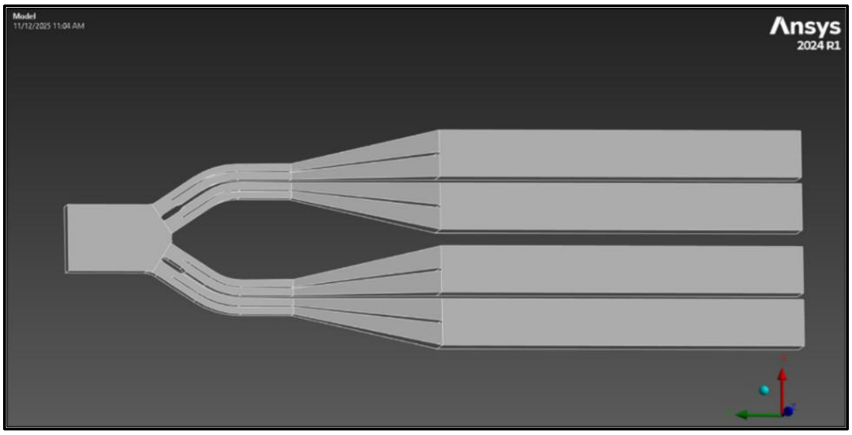

Figure 4. 3D Geometry of MMHPP imported in ANSYS Fluent

A structured rectangular hexahedral grid with a single mesh block was employed to discretize the computational domain. The mesh quality was assessed using the skewness metric, a vital indicator of element quality in Ansys Fluent. A coarse mesh with an average cell size of 0.5m was created to develop the baseline model and evaluate the overall flow dynamics within the computational domain. This initial grid facilitated expedited convergence and offered insights into flow distribution and boundary effects. To enhance solution precision and more effectively capture local flow variations, particularly in areas with significant velocity gradients and near wall boundaries, the mesh was refined to a cell size of 0.15m. The boundary conditions must align accurately with the physical conditions of the problem, as they can significantly influence whether the numerical model outcomes accurately represent actual situations. For the present study, three main boundary types were defined in the computational domain:

a) Inlet Boundary: It is specified as velocity inlet based on the measured discharged from the physical model.
b) Outlet Boundary: It is defined as a pressure outlet.
c) Wall Boundaries: It is applied to all solid surface, with a no – slip condition to simulate realistic fluid wall interaction.

Turbulence is essential to the hydraulic performance of high-Reynolds-number flows, particularly in areas characterized by flow separation, recirculation, and intense shear. The RNG k–ε turbulence model is chosen for the current study. This model is especially appropriate for flows characterized by low turbulence intensity and areas of strong shear.

The simulation outcomes and duration are influenced by several aspects, including time step size, solution method (implicit or explicit), numerical approximations, and convergence criteria. Enhancing runtime and achieving simulation accuracy through the selection of the suitable numerical method is crucial. The solution tab offers several numerical alternatives.

In this study, the coupled algorithm was chosen for pressure-velocity coupling, and a second-order upwind discretization scheme was employed for both the momentum and pressure equations. This combination was selected to enhance solution accuracy, particularly in areas with pronounced velocity gradients and intricate flow patterns, while ensuring numerical stability during the simulation. The outcomes derived from the numerical simulation in Ansys Fluent are examined during the post-processing phase, yielding both quantitative and visual depictions of the flow field. The MMHPP model was then validated with the physical model, before modelling two case studies of Budhi Gandaki Hydroelectric Project (BGHEP).

Two – Dimensional plan and section for case 1 and case 2 of BGHEP from approach culvert to starting of transition of settling basin structure is finalized using AutoCAD and 3D modelling was done in Rhino 3D modelling tool. The geometry developed in .3dm extension in imported to the CFD program.

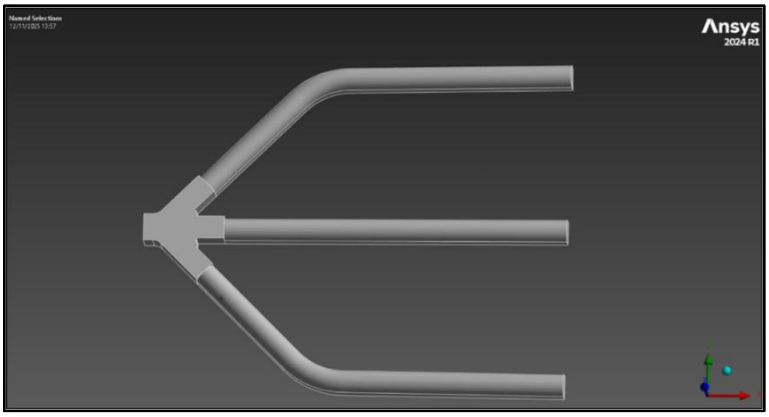

Figure 5. 3D Geometry for case 1 of BGHEP

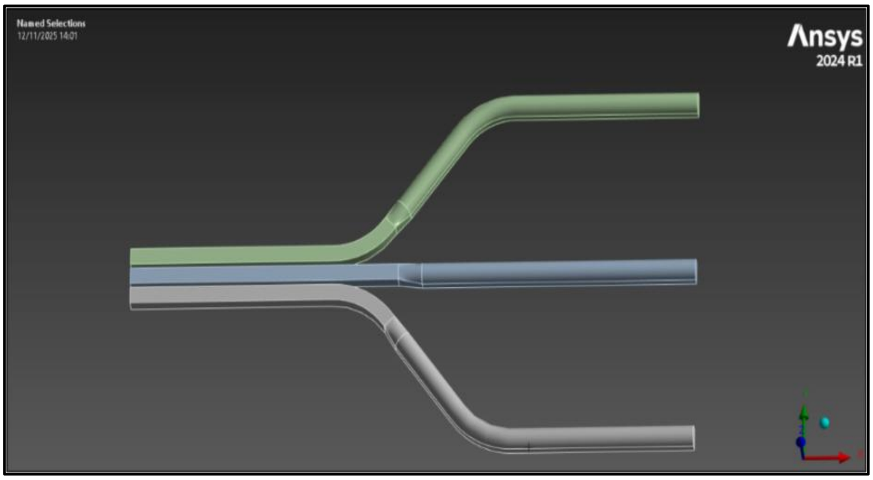

Figure 6. 3D Geometry for case 2 of BGHEP

4. Results and Discussion

4.1 Hydrodynamic Results

This section presents the hydrodynamic results of the settling basin, taking into account the approach flow from the intake. The particular case was modeled with an inflow boundary at the intake, which is standard procedure in settling basin simulations. The velocities and flow distribution in the settling basin were investigated. The output was obtained from the results area of Ansys Workbench. The result acquired for MMHPP was compared with those derived from the physical model test and subsequently validated.

4.2 Results of MMHPP

This scenario was conducted to observe the flow in the settling basin when the inflow boundary is specified at the intake. This is the standard method employed for simulating the settling basin. A steady state was attained at 450 iterations, and the simulation required 3 days and 6 hours for completion.

4.2.1 Flow Field in Settling Basin

There was uneven flow between the bays of the settling basin. The flow velocity in SB4 was the fastest, and the flow velocity in SB1 was the slowest. Also, the velocity was not uniform across the width of bays SB4 and SB1. In SB4, the flow was mostly on the left side of the settling basin, while in SB1, it was mostly on the right side. It was noted that SB2 and SB3 had flow distribution that was almost the same across the board as shown in Figure 7.

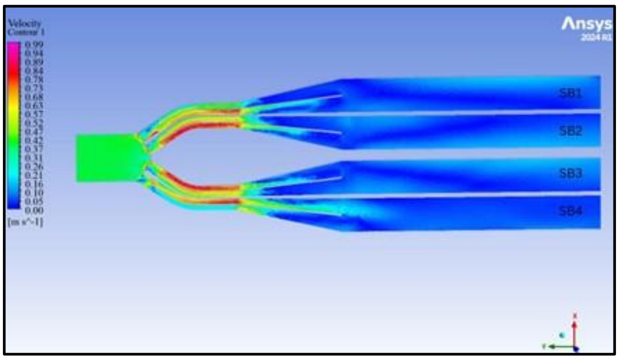

Figure 7. Final Steady State Results of MMHPP

The probe's velocity measurements indicated fluctuations in flow, since velocity changes at different depths of the bay basin at section 1 which is near the transition. Uniform velocity is achieved as the flow progresses toward section 2, located 25 meters downstream from the end of the transition. This is demonstrated by the line graph presented in the Figure 8 and Figure 9**Error! Reference source not found.**.

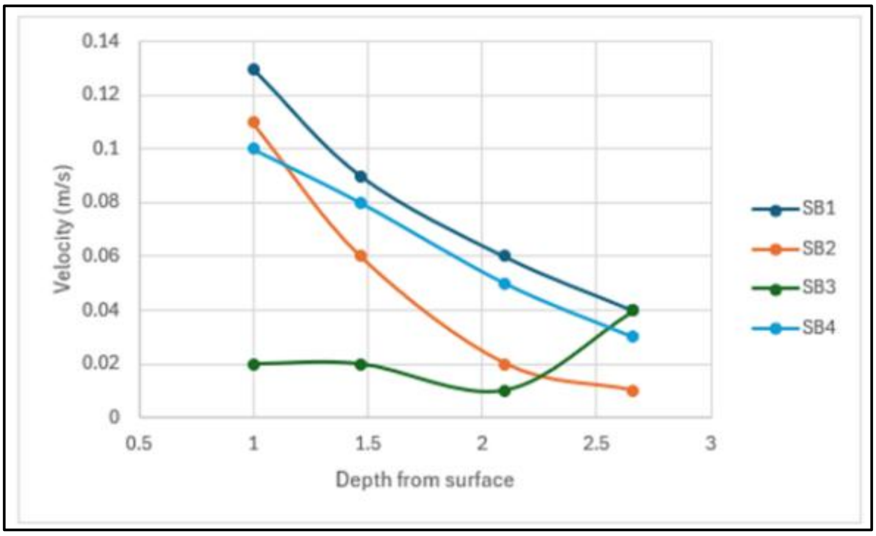

Figure 8. Probe measured velocity at different depth at inlet of transition obtained from Ansys Fluent

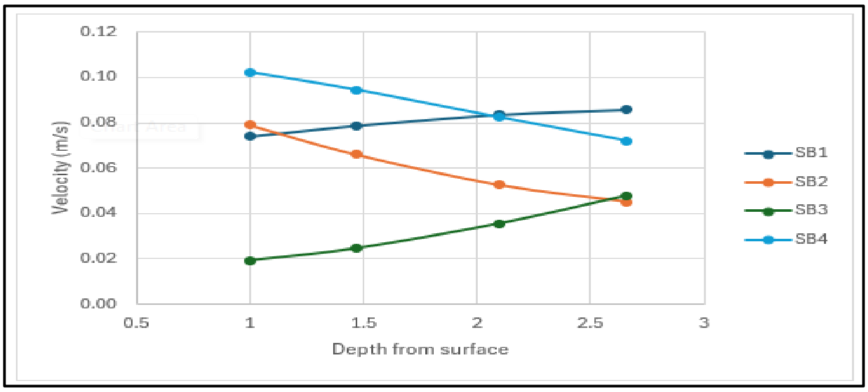

Figure 9. Probe measured velocity at different depth 25m downstream of inlet transition obtained from Ansys Fluent

4.2.2 Flow Turbulent Kinetic Energy along the Settling Basin

Figure 10 illustrates the development of turbulence within the settling basin as determined by the numerical simulation. The turbulence in the basin diminishes from 0.00251 to 0.000001 J/kg. The outermost bays, SB1 and SB4, exhibit stronger turbulence for more than half their length, but the innermost bays, SB2 and SB3, display turbulence only up to half the length of the basin.

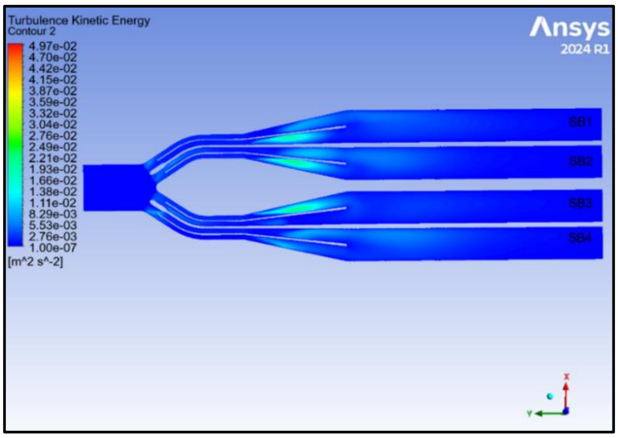

Figure 10. Turbulent Kinetic Energy along the settling basin (MMHPP)

4.2.3 Comparison of Measurements and Simulations

The results from the numerical hydraulic model of the Middle Mewa settling basin are compared to similar ADV data obtained from the physical hydraulic model at Hydro Lab. The model has been compared with the results of the physical model.

4.2.3.1 Comparison of Velocity Profile

Velocities were recorded in the physical model at sections 1 and 2, located 5m and 25m from the intake of the settling basin, as seen in Figure 8**Error! Reference source not found.Error! Reference source not found.** and Figure 9**Error! Reference source not found.**. The velocities were recorded at the midpoint of each bay. The velocity profile at both sections has been calculated from velocity measurements obtained through ADV. The velocity profile for the identical section has been established using velocity data acquired from a probe in the numerical simulation.

Velocity Profile at 5m from the Inlet of the Settling Basin

The velocity profile form at 5 meters from the inflow of the settling basin is illustrated in the Figure 11**Error! Reference source not found.**. At SB1 and SB2, both the models show higher velocities near the surface and lower values towards the bottom. The velocity profiles derived from the physical model is almost similar to the velocity profile derived from numerical model only in SB2. Other three bays have different velocity profile to that obtained from the physical model. The velocity values obtained from the numerical simulation also differs in this section to that of physical model study.

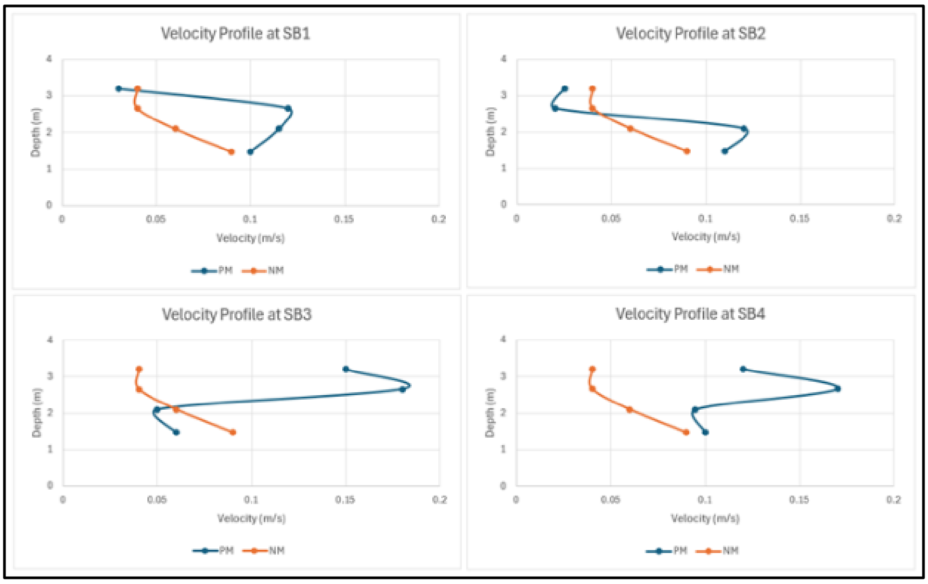

Figure 11. Comparison of velocity profile between Physical and Numerical model at 5m from inlet of settling basin

Velocity Profile at 25m from Inlet of the Settling Basin

The velocity profile at 25 meters from the intake of the settling basin is illustrated in the Figure 12. The velocity profile produced from the physical model closely resembles the velocity profile calculated from the numerical model in each bay. There are minor discrepancies in the velocity values at varying depths between the physical model and the numerical model.

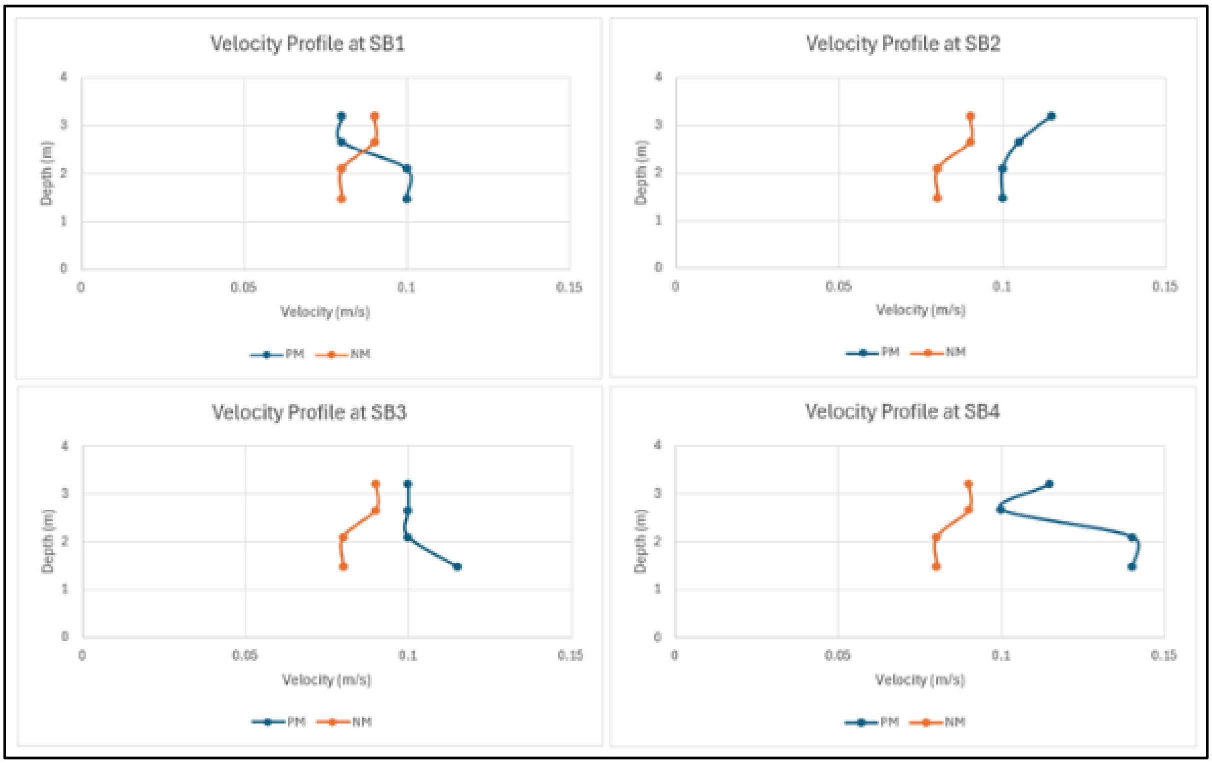

Figure 12. Comparison of velocity profile between Physical and Numerical model at 25m from inlet of settling basin

The aforementioned results indicate that the velocity profile simulated by Ansys Fluent closely approaches that of the physical model in several respects at a distance of 25 meters from the intake of the settling basin. The velocity profile at 5 meters from the inlet of the settling basin differs between Ansys Fluent and the physical model, which can be attributed to the non-uniform flow at that site, as illustrated by the flow pattern in Figure 13.

4.2.3.2 Comparison of Flow Patterns

The study conducted a qualitative analysis comparing the flow patterns of the physical model with the numerical model for the settling basin. The flow pattern from the physical and numerical models is illustrated in Figure 13.

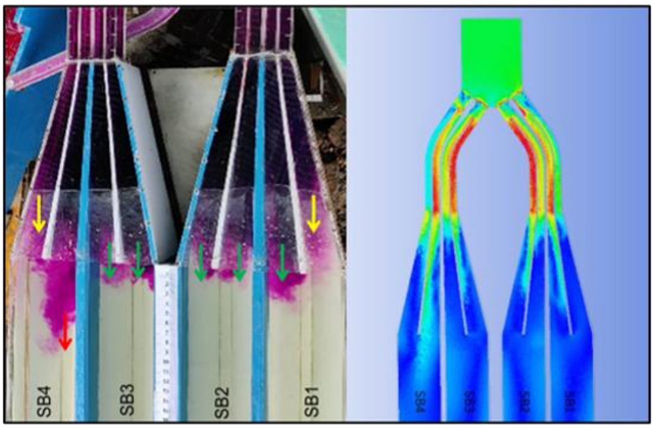

Figure 13. Horizontal flow distribution in the basin Source:(HydroLab, 2022)

It can be seen from Figure 13 that the flow is non uniform near the transition with SB4 having highest velocity and SB1 having least in the numerical and physical model study. Flow field also illustrates shooting flow near the surface of SB4 and certain meter below the surface at SB1. Vertical distribution of the flow field shows that the flow velocity varies with the depth near transition. The variation may have occurred due to turbulence near the transition as observed in Figure 10 . However, the flow seems to be much calmer with almost same velocity at all depth in the bay of the basin after some distance downstream of the transition. More or less uniform flow is achieved at almost 10m downstream of the inlet transition in the physical model as well as numerical model.

4.3 Results of BGHEP

This scenario was conducted to observe the flow in the settling basin when the inflow boundary is specified at the intake. This is the standard method employed for simulating the settling basin. A steady state was attained at 530 iterations, and the simulation required 4 days and 1 hours for completion for case 1, whereas steady state for case 2 was attained at 520 iteration and the simulation required 4 days 1 hours for completion.

4.3.1 Flow Field in Approach Tunnel

For case 1 flow in the mid-section of the approach tunnel tends to travel faster than the other outer tunnel. After the simulation of the numerical model of case 1, the velocity in the outlet of all the approach tunnel was found to be 1.01, 1.15 and 0.98 m/s respectively. This indicates that there is not uniform discharge available in all the part of the settling basin bays. For Case 2 water in all the section of the approach tunnel tends to travel uniformly. After the simulation of the numerical model of case 2, the velocity in the outlet of all the approach tunnel was found to be 1.05 m/s. This indicates that there is uniform discharge available in all the part of the settling basin bays.

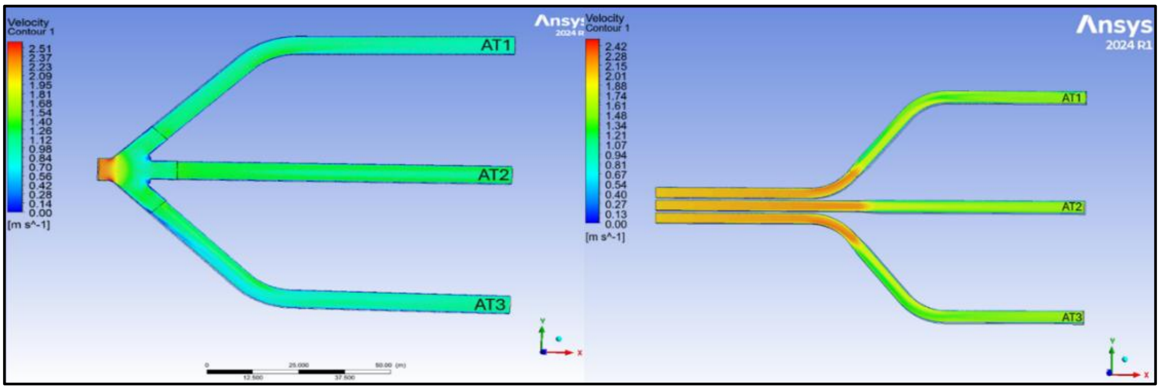

Figure 14. Final Steady State Result of BGHEP for Case 1 (left) and Case 2 (Right)

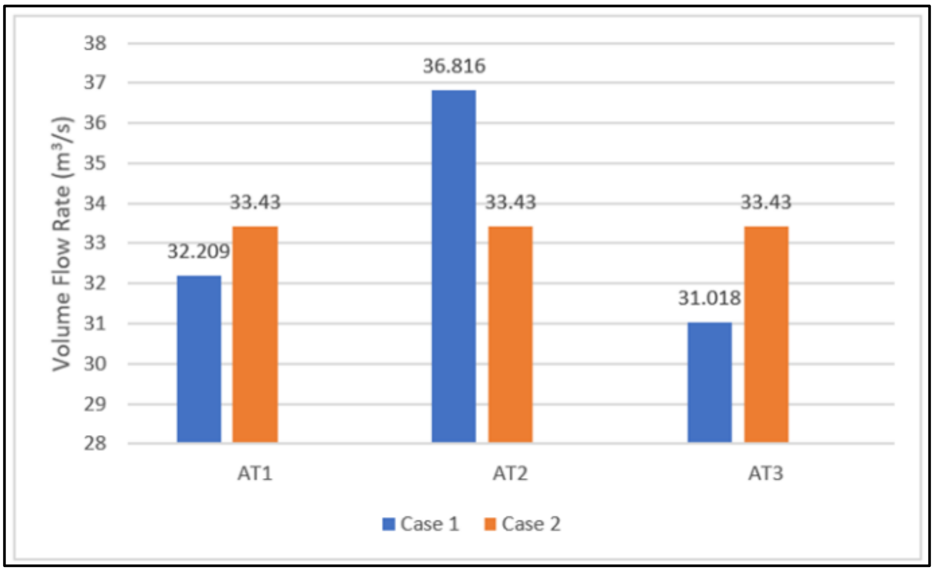

Figure 15. Flow Distribution in 3 bays of Approach Tunnel of BGHEP for Case 1 and Case 2

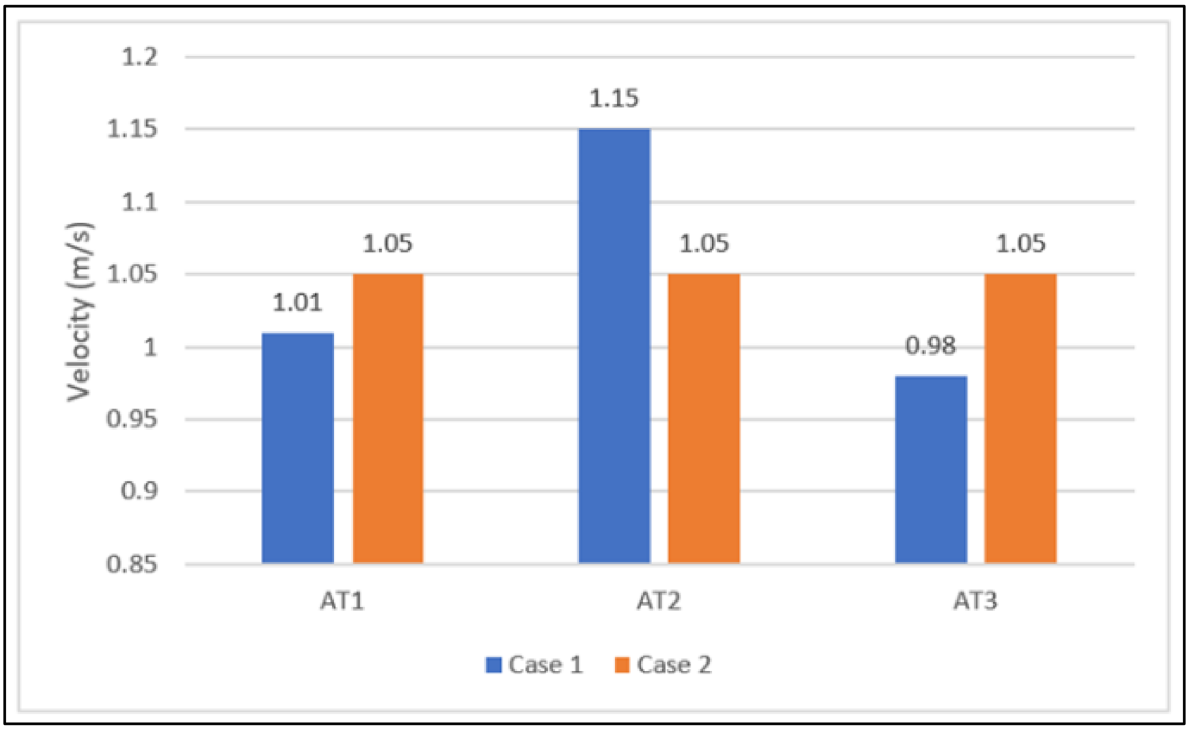

Figure 16. Average Velocity Magnitude in Different bays at the outlet of Approach Tunnel of BGHEP for Case 1 and Case 2

4.3.2 Comparison of Case 1 and Case 2

If we examine both cases, Case 2 shows clear benefits for how water flows through the settling basin. The flow is more stable now, with fewer disruptive input plumes and less overall turbulence. These improvements make it easier for sediment to settle out of the water, thereby fulfilling the main goal of the basin.

Table 1. Comparison of Results (Case 1 vs Case 2)

Aspect	**Case 1**	**Case 2**	**Remarks**
Velocity Uniformity	Uneven distribution among bays	More uniform distribution	Symmetric flow improves efficiency (Xanthos et al., 2011)
Turbulence and Recirculation	Larger vortices, higher turbulence	Lower turbulence, fewer recirculation zones	Improved geometry can cut turbulence by 30–50% (Bruno et al., 2025b)

4.4 Discussions

The discussion demonstrates that the attributes of the approach flow are the most critical determinant of the effectiveness of the settling basin. This aligns with studies on hydropower headworks, which indicate that the primary cause of poorly functioning basins is flow heterogeneity influenced by geometric factors, rather than settling velocity theories (Hillebrand et al., 2017; Paschmann et al., 2022). This study highlights that even minor variations in approach culvert/ tunnel result in significant differences in the internal hydraulics of the basin. Similarly, validated CFD effectively captures subtle yet essential effects. Therefore, the results reinforce the general consensus that approach flow design should be an essential part of settlement basin engineering, especially for Himalayan rivers with a lot of sediment.

This research demonstrates that a validated CFD framework, when calibrated against physical model measurements and applied within appropriate geometric and operational boundaries, constitutes a reproducible approach for analyzing settling basin performance across multiple projects. The present work establishes such a framework, validates it against MMHPP, and extends it to BGHEP. Both Middle Mewa Hydropower Project and Budhi Gandaki Hydroelectric Project are designed as peaking run-of-river schemes with coordinated 6-hour peaking duration. During peaking operation, discharges are released for six hours per day by opening intake gates to divert the design discharge through the headrace system. Beyond operational parallels, MMHPP and BGHEP exhibit substantial geometric similarity in their headworks arrangements. Both projects employ side intake configurations with approach tunnels that transition into settling basins.

Case 2 alignment is superior to case 1 mostly because of improved approach tunnel alignment with the settling basin. This reduces flow separation at inlet and minimize formation of recirculation zones. Moreover. The transition is smooth and gradual allowing flow to reach uniformity at the inlet section. The geometric arrangement in case 2 functions implicitly as a flow-straightening device, even without explicit baffle plates or guide vanes. The refined geometry naturally guides the flow into more parallel streamlines before reaching the basin proper. Based on this study, the configuration similar to case 2 can be adopted as the final design configuration for future hydropower's settling basin approach system.

5. Conclusions

The objective of this study was to analyze the effects of varying approach flow conditions on the hydraulic efficiency of settling basins in hydropower plants. The research established a reliable numerical framework capable of accurately capturing the complex dynamics of approach flow, including velocity gradients, turbulence distribution, through the comparison of empirical measurements from the MMHPP plant with a three-dimensional RANS-based CFD model.

The verified CFD model was used to look at two distinct manners that approach flow could work at the BGHEP headworks. The results illustrate that the type of transition of approach tunnel has a big effect on the flow that enters the settling basin. In Case 1, where the approach channel and the transition were bifurcated from one channel to three channel, there was significant amounts of velocity skewness, visible near-bed recirculation, and high turbulence intensities in the basin. It is known that these characteristics make the effective settling area smaller and contribute to short-circuiting, which makes sediment settling less effective.

Case 2, which featured a smoother transition and a better alignment of the tunnel, led to a significantly more even distribution of velocity at the basin inlet. The turbulence levels were lower, and the entrance jet intensity was lower than in Case 1. The study shows that the primary component affecting hydraulic performance in settling basins is the approach flow state, rather than the basin shape alone. Even if the basin is built well, performance can still be greatly affected by poor approach flow. The study emphasizes how vital it is to develop the headworks with the design of the approach channel in mind.

References

Abilov, R. S. (2024). Research and improvement of the design of a sedimentation tank for hydropower and irrigation. Naukovij Žurnal «Tehnìka Ta Energetika», 15(3), 60–72. https://doi.org/10.31548/machinery/3.2024.60

ANSYS Inc. (2006). ANSYS fluent Theory Guide.

Baluch, K., Park, H.-J., Ji, K., & Baluch, S. Q. (2024). 3D Numerical Model Validation Using Experimental Test Data for Patrind Hydropower Project. Water, 16(2), 360. https://doi.org/10.3390/w16020360

Bhandari, R., Neupane, N., Shrestha, S., Chauhan, D., Pokharel, D., & Xue, W. (2025). Environmental and social considerations in hydropower development in the South Asian Himalayas: a NEXUS perspective. Sustainability Nexus Forum, 33(1), 19. https://doi.org/10.1007/s00550-025-00578-w

Bishwakarma, M. (2015). Settling Basin Design Criteria and Trap Efficiency Computation Methods. Civil Engineering Voice, Vol 1, 2015.

Bor, A., Szabo-Meszaros, M., Vereide, K., & Lia, L. (2024). Application of Three-Dimensional CFD Model to Determination of the Capacity of Existing Tyrolean Intake. Water, 16(5), 737. https://doi.org/10.3390/w16050737

Bouvard, M. (1992). Mobile barrages and intakes on sediment transporting rivers. Balkema.

Bruno, P., Bruno, F., Di Bella, G., Napoli, E., & De Marchis, M. (2025a). CFD study on the effect of the baffles geometry in sedimentation efficiency in wastewater treatments through Large Eddy Simulations. Journal of Environmental Management, 373, 123536. https://doi.org/10.1016/j.jenvman.2024.123536

Bruno, P., Bruno, F., Di Bella, G., Napoli, E., & De Marchis, M. (2025b). CFD study on the effect of the baffles geometry in sedimentation efficiency in wastewater treatments through Large Eddy Simulations. Journal of Environmental Management, 373, 123536. https://doi.org/10.1016/j.jenvman.2024.123536

C. S. Thaxton, J. Calantoni, & R. A. McLaughlin. (2004). HYDRODYNAMIC ASSESSMENT OF VARIOUS TYPES OF BAFFLES IN A SEDIMENT RETENTION POND. Transactions of the ASAE, 47(3), 741–749. https://doi.org/10.13031/2013.16106

Camp, T. R. (1946). Sedimentation and the design of settling tanks. Transactions of the American Society of Civil Engineers, 111, 895–912.

Chen, C. (1975). Design of sediment retention basins. Proceedings of the national symposium of urban hydraulic sediment control.

Chitrakar, S., & Neopane, H. (2019, November). River sedimentation challenges for hydropower development in Nepal: A perspective review.

Dick, R. I. (1981). Sedimentation Since Camp. Boston Society of Civil Engineers Section, ASCE, 68(2), 199–236.

Dulal, S. (2020). Numerical Simulation of Flow Through Settling Basin: A case study of Budhi-Ganga Hydropower Project (BHP). International Journal of Engineering Research And, V9(07). https://doi.org/10.17577/IJERTV9IS070407

El-Eraqui, M., EL-Nahhas, K., & Saad Eddin, S. (2020). Investigation of Rectangular Settling Tanks Performance in Drinking Water Treatment Plants. Bulletin of the Faculty of Engineering. Mansoura University, 40(1), 11–21. https://doi.org/10.21608/bfemu.2020.96274

Gebrelibanos, K. G., Vereide, K., Weldemariam, S. A., Bor, A., Tesfay, A. H., & Lia, L. (2025). Hydraulic Scale Modeling of Pressurized Sediment Laden Flow. Water, 17(13), 1970. https://doi.org/10.3390/w17131970

Goula, J. P., Elena, M., Dolan, J., & Aydiner, C. (2008). A CFD methodology for the design of sedimentation tanks in potable water treatment: Case study: The influence of a feed flow control baffle. Chemical Engineering Journal, 140(1–3), 152–164. https://doi.org/10.1016/j.cej.2007.09.018

Gualtieri, C., Angeloudis, A., Bombardelli, F., Jha, S., & Stoesser, T. (2017). On the Values for the Turbulent Schmidt Number in Environmental Flows. Fluids, 2(2), 17. https://doi.org/10.3390/fluids2020017

Hillebrand, G., Klassen, I., & Olsen, N. R. B. (2017). 3D CFD modelling of velocities and sediment transport in the Iffezheim hydropower reservoir. Hydrology Research, 48(1), 147–159. https://doi.org/10.2166/nh.2016.197

Hirom, K., & Devi, T. T. (2022). Application of Computational Fluid Dynamics in Sedimentation Tank Design and Its Recent Developments: a Review. Water, Air, & Soil Pollution, 233(1), 22. https://doi.org/10.1007/s11270-021-05458-9

Huang, B., David, Z., & Fu, J. (2013, November). A Study on Flow Field Induced by Hydropower Intakes.

HydroLab. (2022). Physical Hydraulic Model Study of the Headworks of Middle Mewa Hydropower Project (73.5 MW) Draft Final Report.

Imam, E., McCorquodale, J. A., & Bewtra, J. K. (1983). Numerical model of secondary clarifiers. Journal of the Environmental Engineering Division, 109(2), 349–368. https://doi.org/10.1061/(ASCE)0733-9372(1983)109:2(349)

International Hydropower Association (IHA). (2022). Net zero. https://www.hydropower.org/net-zero

Iyer, C. J., & James, E. J. (2023). Indispensability of Model Studies in the Design of Settling Basins of Hydropower Projects in River Basins with High Sediment Yield. In P. V Timbadiya, P. L. Patel, V. P. Singh, &

B. Barman (Eds.), Fluid Mechanics and Hydraulics. HYDRO 2021 (Vol. 314). Springer. https://doi.org/10.1007/978-981-19-9151-6_30

Khan, L. A., Wicklein, E. A., Rashid, M., Ebner, L. L., & Richards, N. A. (2004). Computational fluid dynamics modeling of turbine intake hydraulics at a hydropower plant. Journal of Hydraulic Research, 42(1), 61–69. https://doi.org/10.1080/00221686.2004.9641184

Lipeme Kouyi, G., Arias, L., Barraud, S., & Bertrand-Krajewski, J.-L. (2010, November). CFD Modelling of flows in a large stormwater detention and settling basin.

Metcalf & Eddy, Inc., Tchobanoglous, G., Burton, F. L., & Stensel, H. D. (2003). Wastewater Engineering: Treatment and Reuse (4th ed.). McGraw-Hill.

Mewa Developers. (2019). Middle Mewa Hydropower Project(49MW) Taplejung, Nepal Updated Feasibility Study Volume-I Main Report.

Michael, Y., & Tilaye, R. (2010). Numerical Modeling for Settling Basin Design.

Mishra, M. (2023, November). Sediment management in hydropower plants – a case study for Himalayan region of India.

Mohammad, M. E., Al-Ansari, N., Knutsson, S., & Laue, J. (2020). A Computational Fluid Dynamics Simulation Model of Sediment Deposition in a Storage Reservoir Subject to Water Withdrawal. Water, 12(4), 959. https://doi.org/10.3390/w12040959

Morris, G. L. (2016). Sediment Management at Run-of-River Headworks. Extending the Life of Reservoirs, 127.

Morris, G. L., & Fan, J. (1998). Reservoir sedimentation handbook: design and management of dams, reservoirs, and watersheds for sustainable use.

Nøvik, H., Dudhraj, A., Olsen, N. R., Bishwakarma, M. B., & Lia, L. (2014). Numerical Modeling of Nonuniform Flow in Settling Basins. Hydro Nepal: Journal of Water, Energy and Environment, 14, 27–35. https://doi.org/10.3126/hn.v14i0.11251

Olen, S. M., Bookhagen, B., Hoffmann, B., Sachse, D., Adhikari, D. P., & Strecker, M. R. (2015). Understanding erosion rates in the Himalayan orogen: A case study from the Arun Valley. Journal of Geophysical Research: Earth Surface, 120(10), 2080–2102. https://doi.org/10.1002/2014JF003410

Pandey, R., Shrestha, R., Bhattarai, N., & Dhakal, R. (2023). Problems identification and performance analysis in small hydropower plants in Nepal. International Journal of Low-Carbon Technologies, 18, 561–569. https://doi.org/10.1093/ijlct/ctad043

Paschmann, C. (2018). Design Optimization of Desanding Facilities for Hydropower Schemes. https://api.semanticscholar.org/CorpusID:115678396

Paschmann, C., Vetsch, D. F., & Boes, R. M. (2022). Design of Desanding Facilities for Hydropower Schemes Based on Trapping Efficiency. Water, 14(4), 520. https://doi.org/10.3390/w14040520

Pokhrel, A. (2024). Monitoring sediment levels in hydropower: Ensuring dam safety and plant efficiency. https://www.urjakhabar.com/en/news/0807243910

Rodi, W. (2017). Turbulence Modeling and Simulation in Hydraulics: A Historical Review. Journal of Hydraulic Engineering, 143(5). https://doi.org/10.1061/(ASCE)HY.1943-7900.0001288

Shahrokhi, M., Rostami, F., Md Said, M. A., & Syafalni. (2013). Numerical modeling of baffle location effects on the flow pattern of primary sedimentation tanks. Applied Mathematical Modelling, 37(6), 4486–4496. https://doi.org/10.1016/j.apm.2012.09.060

Shinbein, M. (2021). Physical Modeling of Rivers in the Hydraulics Laboratory (Issue ST-2021-21200).

Shrestha, B. V. (2012). 3D Numerical Investigation on Settling Basin Layout: A Case Study on Mai Khola Hydropower Project, Nepal.

Simanjuntak, T. D. Y. F., Boeriu, P., & Roelvink, J. A. D. (2009). Consideration on the sedimentation process in a settling basin. Journal of Hydrology and Hydromechanics, 57(1), 16–25. https://doi.org/10.2478/v10098-009-0002-4

Stamou, A. I. (1995). Modelling of settling tanks – a critical review. Transactions on Ecology and the Environment, 7, 305–312.

Thapa, B., Shrestha, R., Dhakal, P., & Thapa, B. (2004). Sediment in Nepalese Hydropower Projects.

Times Energy. (2024). Updated Detailed Feasibility Study of Budhi Gandaki Hydroelectric Project (341 MW) Updated Detailed Feasibility Study Report.

Vanoni, V. A. (1975). Sedimentation Engineering (V. A. Vanoni, Ed.). American Society of Civil Engineers.

Vasquez, J., Hurtig, K., & Hughes, B. (2014). Computational Fluid Dynamics (CFD) Modeling of Run-of-River Intakes. https://www.flow3d.com/wp-content/uploads/2014/08/Computational-Fluid-Dynamics-CFD-Modeling-of-Run-of-River-Intakes.pdf

Vieira Lucchese, L., & Camano Schettini, E. (2016, November). Turbulence Analysis of a Rectangular Sedimentation Basin Using Numerical Simulation.

Wilcox, D. C. (1998). Turbulence Modeling for CFD (2nd ed.). DCW Industries.

Xanthos, S., Gong, M., Ramalingam, K., Fillos, J., Deur, A., Beckmann, K., & McCorquodale, J. A. (2011). Performance Assessment of Secondary Settling Tanks Using CFD Modeling. Water Resources Management, 25(4), 1169–1182. https://doi.org/10.1007/s11269-010-9620-1

Zamani Nouri, A., & Heydari, M. M. (2017). Performance and flow field assessment of settling tanks using experimental and CFD modeling. Membrane Water Treatment, 8(5), 423–435. https://doi.org/10.12989/mwt.2017.8.5.423