International Journal on Engineering Technology (InJET)

www.kec.edu.np/journal
Volume1, issue no: 1, Nov 2023, Page 153-165
Received Date: Aug 1, 2023
Accepted Date: Oct 9, 2023
ISSN: 3021-940X (print)

Numerical Model Scenario Analysis of Stilling Basin: A Case Study of Tanahu Hydropower Project (140 MW)

Postaraj Khadka[1 *], Suman Rai[2]

[1]*Institute of Engineering, Pulchowk Campus, Pulchowk, Lalitpur, Nepal, 074mshpe010.postaraj@pcampus.edu.np*
[2]*GoN, Ministry of Urban Development, Department of Local Infrastructure, Kathmandu, Nepal, ersumansampang@gmail.com*

Abstract

The difficulty of experimental methods to modify the complex model motivated the researchers to explore alternative solutions. Tanahu Hydropower Project is a storage-type hydropower project. It has an installed capacity of 140 MW and 140 m high concrete dam along with chute-type stilling basin followed by a complex topography. Computational fluid dynamic (CFD) model is a numerical approximation of partial differential equations. It has been widely used to simulate the fluid flow. In this study, a fluid flow is simulated through the stilling basin using the numerical model for different return period floods. The model's predictions for flow parameters are validated with the results taken from the 1:60 scaled physical models for the same project. The results regarding the flow velocities and water surface level are within 30% and 1.92 m accuracy respectively. The validated model is run for the three modification cases: i) by opening only two of the three spillway gates, ii) by decreasing the depth of the stilling basin, and iii) by decreasing the length of the basin, aiming to recommend the best alternative solution for the effective dissipation of the high kinetic energy of flow from the 140 m high dam. The results reveal that the base case model is the best solution compared to these three modified cases to pass the flood effectively. This study concludes that the CFD model is the effective alternative tool to analyze the fluid flow problems even in the complex geometry and is recommended to use for the design and modification processes.

Keywords: Numerical Analysis, Stilling Basin, Computational Fluid Dynamics, Tanahu Hydropower Project, Roller Bucket

1. Introduction

Tanahu Hydropower Project is located 150 km west of Kathmandu on the Seti River near Damauli of Tanahu District in Gandaki Province. It is a storage-type hydropower project having an installed capacity of 140 MW with an estimated average annual energy generation of 587.7 GWh (years 1-10) and 489.9 GWh (years 11 onwards). The project has a 140 m high concrete gravity dam with a chute-type spillway. The spillway consists of an ogee-type profile at the upper part filled by a curve to join the bucket-type energy dissipator at the toe of the dam-the downstream of the dam is attached to the surrounding complex topography. An auxiliary dam is attached at the downstream side from the energy dissipator to form a plunge pool that reduces the scouring effect.

Many researchers have done several studies regarding the flow through the stilling basins and hydraulic structures. A study conducted for flow characteristics in the hydraulic jump stilling basin with a numerical model (Cook and Richmond, 2001). They found accuracy within 25%. Another study conducted for a 3-D simulation of flow in a stilling basin with a baffle and found the results within standard deviation (Cook et al., 2002). A study of a numerical simulation conducted for a stilling basin of multi-horizontal submerged jets (Chen et al., 2010). The comparative results with different model experiments showed that flow parameters such as water depth, pressure distribution, and velocity profile were in good agreement. The hydraulic jump in the

**Corresponding Author*

convergence United States Bureau of Reclamation (USBR) II stilling basin was conducted using Flow-3D numerical model (Babaali et al., 2015). The predicted flow pattern in the stilling basin was in good agreement with the general flow pattern. The results obtained from numerical simulation using two turbulent models: k-ε and RNG models were compared and found that RNG turbulent model gave the better results. The CFD models were also able to capture the flow behavior in the Multi-Horizontal Submerged Jets (MHSJ) stilling basins (Bayón et al., 2019). The addition of the adverse slope to end of United States Bureau of Reclamation (USBR) II stilling basin was done using the numerical model software Flow-3D (Babaali et al., 2019). The results from the numerical model and 1:40 scaled physical model was in good agreement. The analysis of the hydraulic jump occurring in a typified USBR type II stilling basin was conducted using the numerical and experimental modeling approach (Macián-Pérez et al., 2020). A physical model was built by reducing the scale and the same size model was prepared for CFD simulation also. From the both models, it was observed that models were able to represent the flow characteristics such as hydraulic jump shape, velocity profiles, and pressure distributions.

All of the previous research was conducted for a regular shape stilling basin. None of them were irregular shapes. This study attempted to conduct the numerical study for an irregular shape stilling basin having the complex topography.

Due to the flow over the high dam, there will be scouring problems downstream. To reduce the effect, different energy dissipators be recommended. In the current study, the complex topography at the dam's downstream side enhances the scouring problem and makes it very difficult for smooth flow operation. Historically, physical models were used to analyze the flow through hydraulic structures, but it has many disadvantages like high construction and simulation cost, long simulation time and scaling effects (Briggs, 2013). Due to the growing information technology, high processing computers can give better results through different numerical models. In this study, the flow is passed through the stilling basin using the Computational Fluid Dynamics model, compared its results with the physical models, and the scenario analysis is conducted with the three different modification cases.

2. Study Area

The Tanahu hydropower project lies at 150 km west of Kathmandu on the Seti River near Damauli of Tanahu District of Gandaki Province of Nepal. The project was planned to be developed by Nepal Electricity Authority (NEA), a Utility Company of the Government of Nepal. Electric Power Development Co. LTD. (J-POWER), Japan, has been undertaken the Detailed Engineering Study (DES) of this project under the financial assistance of the Asian Development Bank (ADB) (Thapa and Bogati, 2012). The project has the following salient features:

Table 1. Salient features of the project

Items	Quantity
Design Discharge	151.2 m^3/s
Installed Capacity	140 MW
Number of Units	Two
Effective Head	121.55 m
Total Energy	587.7 GWh (1-10 year)
	489.9 GWh (11^{th} year onwards)
Reservoir Length	18 km
Reservoir Area	7.26 km^2
Dam Type	Concrete Gravity Dam

Length at Crest	175 m
Height of Dam	140 m
Spillway Type	Chute Type Dated Spillway
Design Flood	7,377 m^3/s
Number of Gates	Three
Energy Dissipator	Roller Bucket Type

The project is capable of supplying a peak power for minimum of 6 hours daily in dry season. The dam site is at 1 km upstream and powerhouse at 3 km downstream of confluence of the Seti and Madi River. The stilling basin consists of roller bucket type energy dissipator with radius of 15.0 m, sill height 6.4 m and releasing angle of 30°. The dam has three spillway gates leading the profile to join with the roller bucket at the toe of the dam and stilling basin downstream. The auxiliary dam is at about 178 m downstream measuring along the center line from the toe of the dam. The return period floods corresponding to 2, 10, 100, 500, and 10000 years are 1000, 2000, 3000, 4000, and 5500 m^3/s respectively, where the probable maximum flood (PMF) is 7,377 m^3/s.

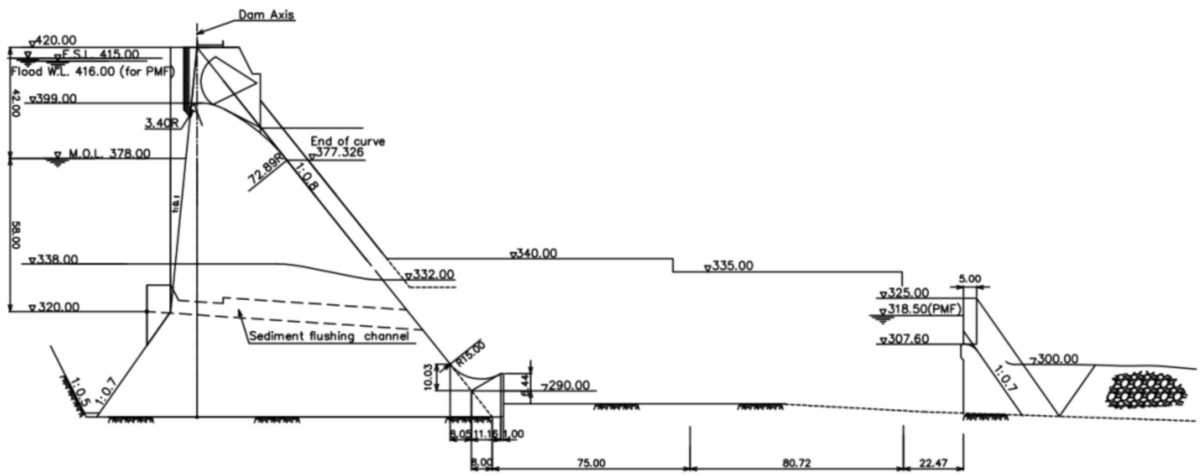


Figure 1. Longitudinal section of stilling basin (Thapa and Bogati, 2012)

3. Numerical Model and Methodology

Computational Fluid Dynamics (CFD) is a computer software based on numerical technique to analyze the problems related to hydraulic fluid flows (Versteeg and Malalasakera, 2007). The CFD simulations predict the fluid flow field by solving the Navier-Stokes' energy and mass conservation equations over a region of interest. It has the advantage of calculating the mechanistic data of the complete fluid flow within the domain. .

The continuity equation is given as:

$$V_f \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho u A_x) + \frac{\partial}{\partial y}(\rho v A_y) + \frac{\partial}{\partial z}(\rho w A_z) = 0$$

The momentum equations are presented as:

$$\frac{\partial u}{\partial t} + \frac{1}{V_f}(uA_x \frac{\partial u}{\partial x} + vA_y \frac{\partial u}{\partial y} + wA_z \frac{\partial u}{\partial z} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + G_x + f_x$$

$$\frac{\partial v}{\partial t} + \frac{1}{V_f}(uA_x \frac{\partial v}{\partial x} + vA_y \frac{\partial v}{\partial y} + wA_z \frac{\partial v}{\partial z} = -\frac{1}{\rho}\frac{\partial p}{\partial y} + G_y + f_y$$

$$\frac{\partial w}{\partial t} + \frac{1}{V_f}(uA_x \frac{\partial w}{\partial x} + vA_y \frac{\partial w}{\partial y} + wA_z \frac{\partial w}{\partial z} = -\frac{1}{\rho}\frac{\partial p}{\partial z} + G_z + f_z$$

where, V_f indicates fraction of open volume, ρ indicates fluid density, (u, v, w) indicate velocity components along x-, y- and z- directions, A indicates fraction of open level, G indicates mass acceleration and f indicates viscosity acceleration

The entire region of interest is discretized into cells. The volume averaged conservation equations of mass, momentum and energy are then solved over this domain. The accuracy of the solution generally increases with the increase in the number of cells but smaller grid size results in longer computational time. After reviewing the literature and previous data, the general flowchart of this study is summarized as shown in figure 2.

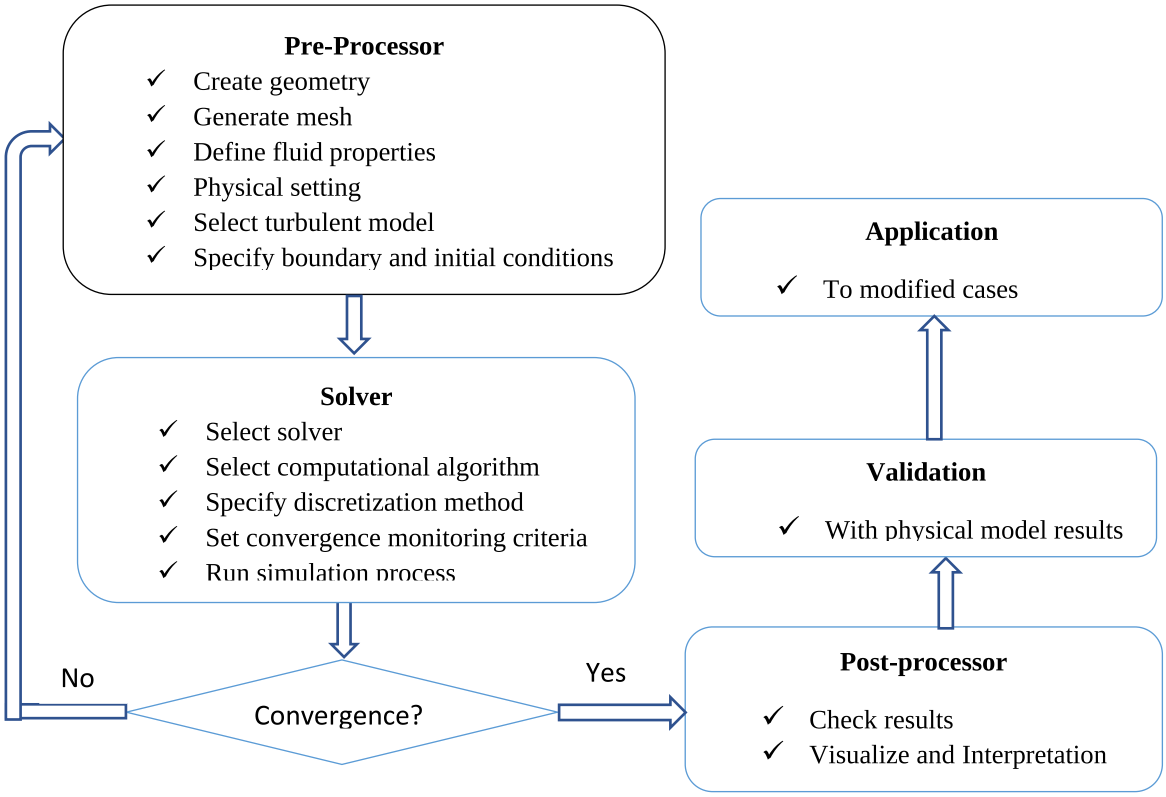


Figure 2. Flowchart of CFD model

The CFD model mainly consists of three stages:

3.1. Pre-processor

It consists of the following procedures:

- Defining the geometry of the computational domain
- Generating the grids by dividing the domain into the number of small sub-domains
- Defining the fluid properties and other physical, chemical, and biological parameters
- Specifying the appropriate initial and boundary conditions
- Specifying the simulation parameters: time step size, simulation duration, result output intervals, and other numeric

Flow-3D software is used to run the model that uses the three-dimensional Navier-Stokes equations with continuity and momentum equations. From the information and drawing provided by Tanahu Hydropower Limited, the three-dimensional geometry of the model is drawn in AutoCAD-3D. The model is the imported

into Flow-3D software where the system's domain is meshed with Cartesian mesh with 16 different mesh blocks with a mesh size of 0.75 m. The domain covers 150 m upstream to 340 m downstream from the dam axis. The volume flow rate is taken as the upstream boundary condition and outflow as the downstream boundary condition.
The upstream and downstream boundary conditions are set to the volume flow rate and outlet respectively.

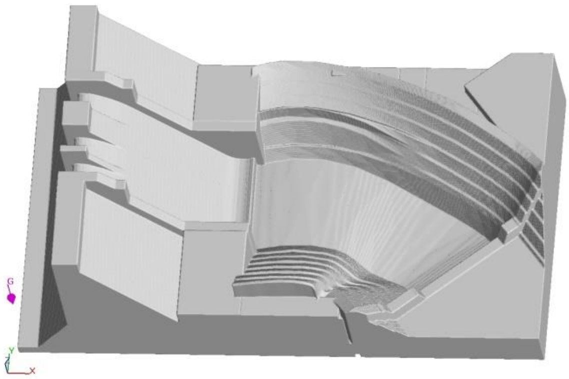


Figure 3. 3-dimensional model

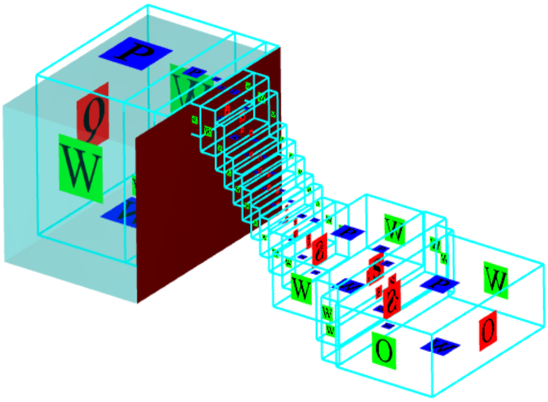


Figure 4. Boundary conditions

3.2. Solver

It consists of the following procedures:

- Integrating the governing equations of fluid flow over all the domain control volume
- Discretizing by converting the resulting integral equations to algebraic equations
- Solving the algebraic equations by iterative method

Among different available turbulent models, the RNG $k - \varepsilon$ turbulent model is used in this study. It gives better results in the free surface modeling (Sabbagh-Yazdi etal., 2007).
In this study, an incompressible Newtonian fluid is taken for all the simulations. In time-step control, time step is set to 10^{-9} sec so that it could not able to fall below minimum time step to neglect the termination of the simulation process due to stability criteria. Maximum number of iterations in the convergence control for pressure is set to 1000 so that it helps to converge the solution.

3.3. Post-processor

It includes the results and output from the model in the formats: domain geometry and grid display, vector plots, contour plots, 2-D and 3-D surface plots. The model is run for 2, 10, 100, 500, and 10,000 years return period and probable maximum flood over the spillway towards the stilling basin to observe the flow pattern and parameters. The results obtained from the numerical model are verified with the results of the physical model for the base case. The scenario study is conducted by modifying the model by i) opening only two spillway gates, ii) decreasing the depth of the stilling basin, and iii) shifting the location of the auxiliary dam.

4. Results and Discussion

The flood discharges are passed through the stilling basin to observe the flow patterns and flow behavior by using the numerical simulation. The obtained results are validated with the results of the physical model study. The water profile and velocity of the flow obtained from the numerical model are validated with that from the physical model for the different return period floods. It is observed that the flow profile produces a nappe that closely agreed with the shape of the spillway. The flow changes rapidly from sub-critical to critical when it took place from the spillway until it met the roller bucket at the toe of the dam. Through the spillway, the maximum flow velocities are observed at the middle portion rather than the sides of the basin, which will reduce the chances of scouring at the sides by the high floods. For each design flood, similar phenomena are observed. The performance of the stilling basin is found to be satisfactory for the design floods up to the 500-years return period. The performance is well represented by this numerical model.

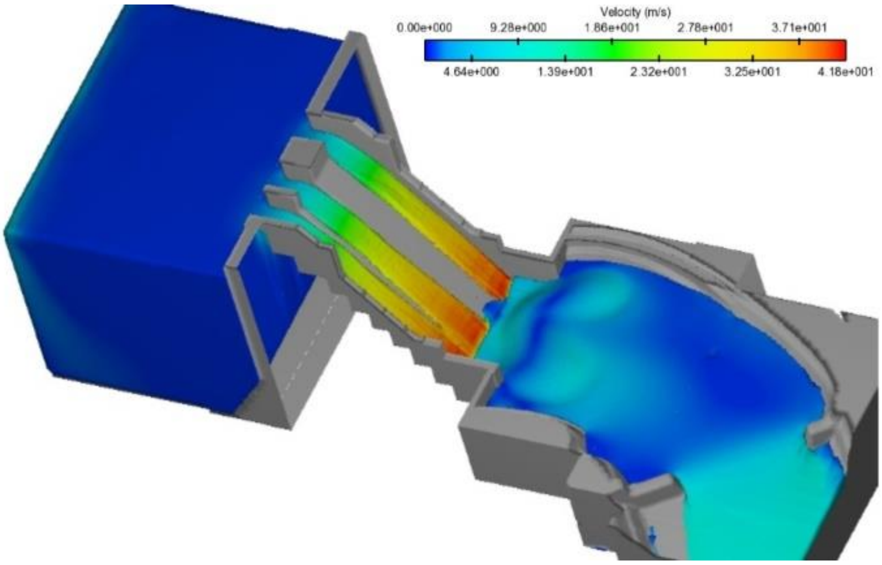


Figure 5. Velocity contour for 4000 m^3/s (500 year return period flood)

The measurements of the water surface profile are taken by averaging the results for 60 seconds time interval after the stable flow condition. The differences in the water surface levels between the numerical and 1:60 scaled physical models are within 1.66 m, 0.92 m and 1.92 m for 10, 100, and 500 years return period floods respectively.

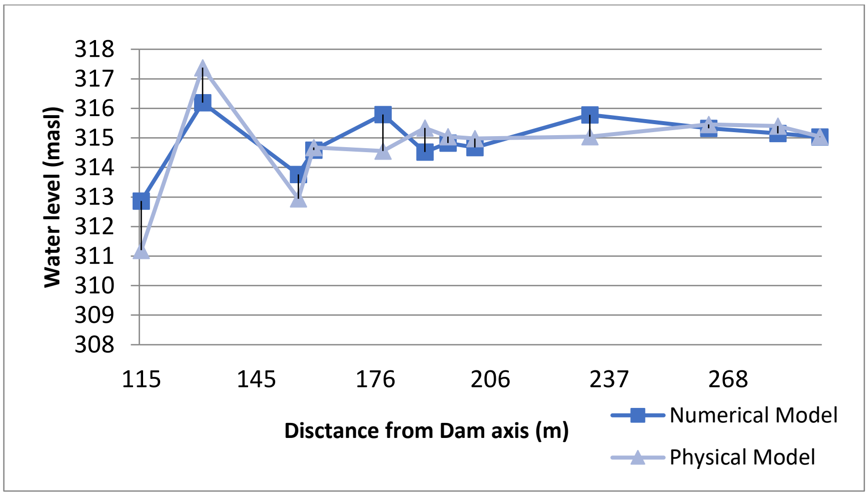


Figure 6. Water surface profile along main course for 2000 m^3/s

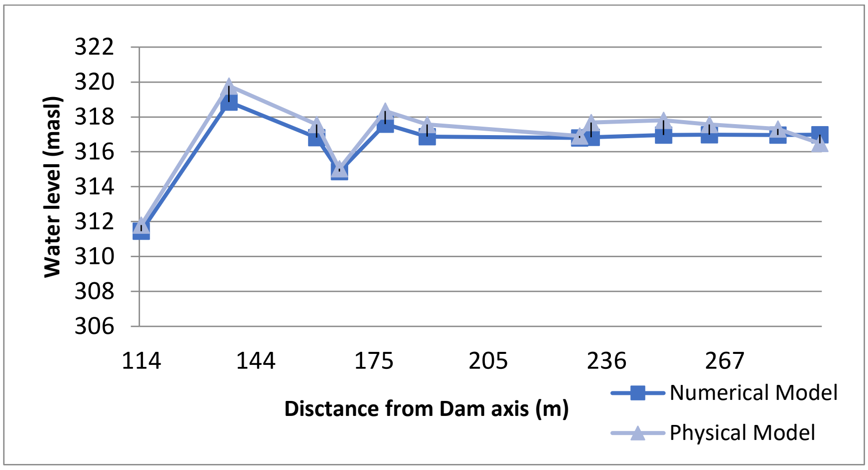


Figure 7. Water surface profile along main course for 3000 m^3/s

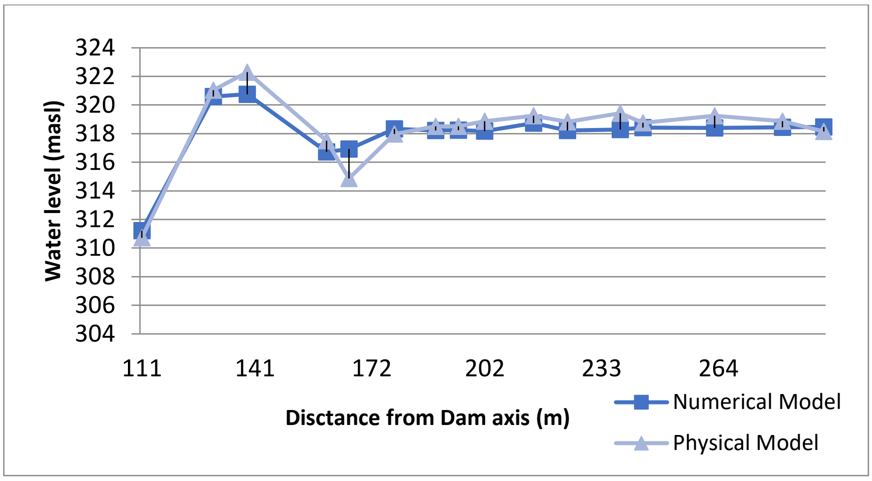


Figure 8. Water surface profile along main course for 4000 m^3/s

The velocity distributions at the three different levels: 293.00, 305.00, and 309.00 masl are measured by Flow Sight in the Flow-3D for 10, 100 and 500 year return period floods. The measurements of the flow velocity are taken by averaging the results for 60 second time interval after the stable flow condition. The measured data are then compared to that of physical model for the same measurement points as shown in the figure 9.

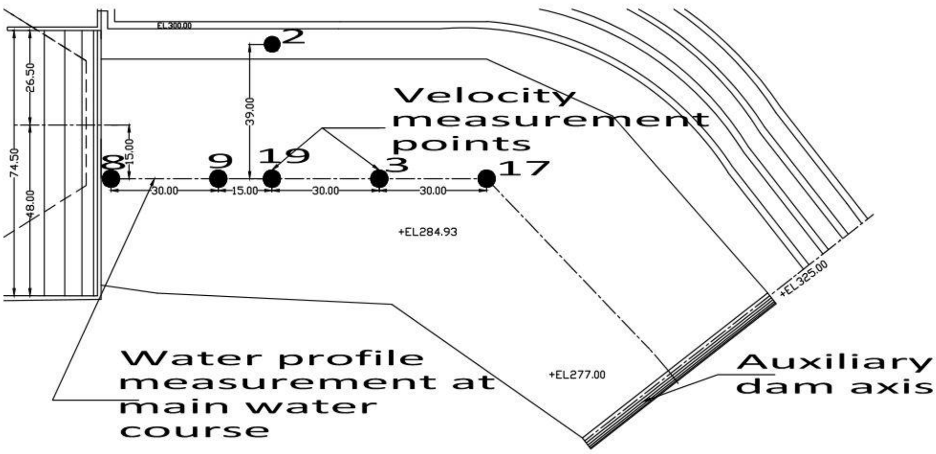
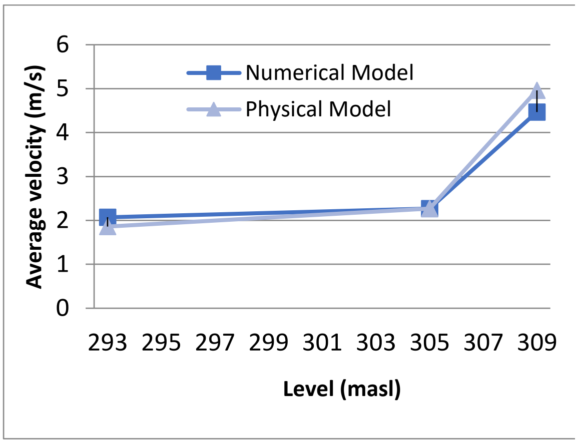
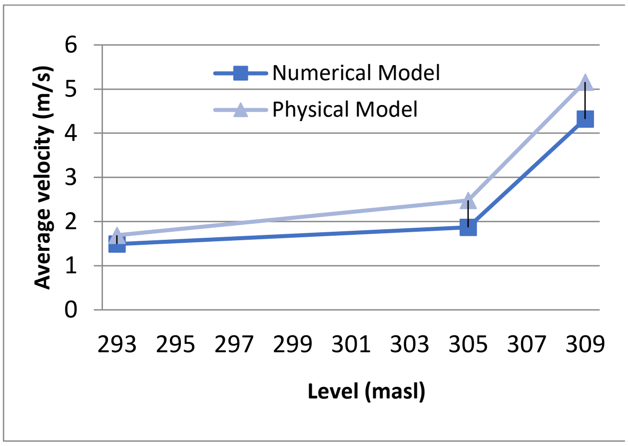


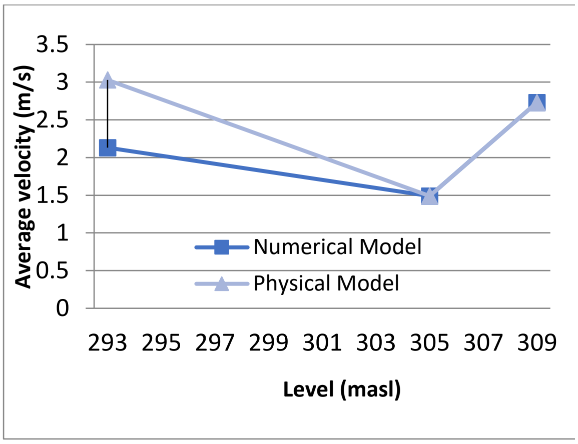
Figure 9. Measurement points for flow velocity (Thapa and Bogati, 2012)



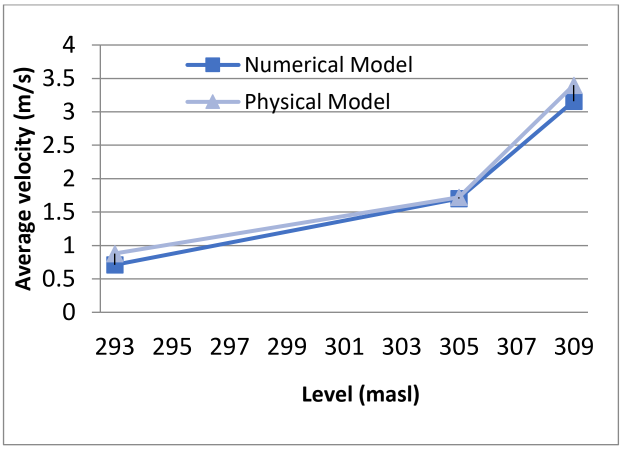
a) point 9



b) point 19

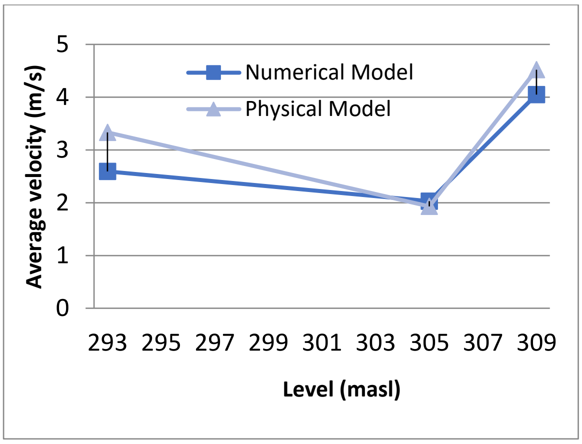


c) point 2

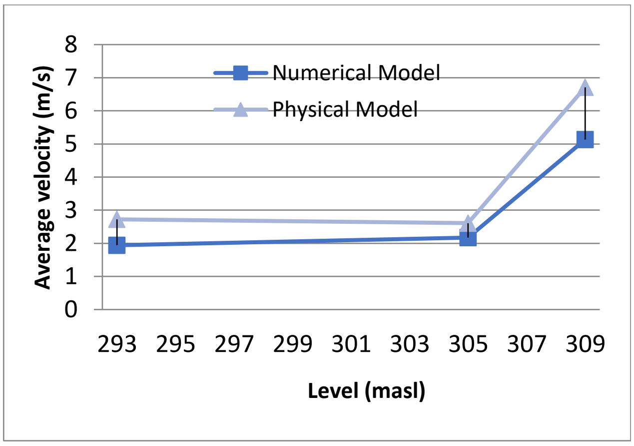


d) point 3

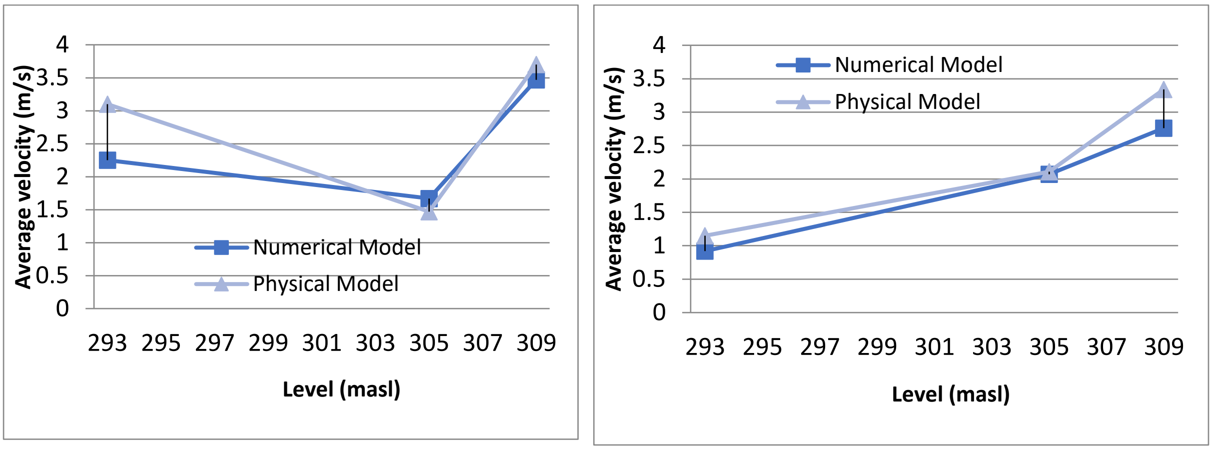
Figure 10. Average velocity for 2000 m^3/s



a) point 9

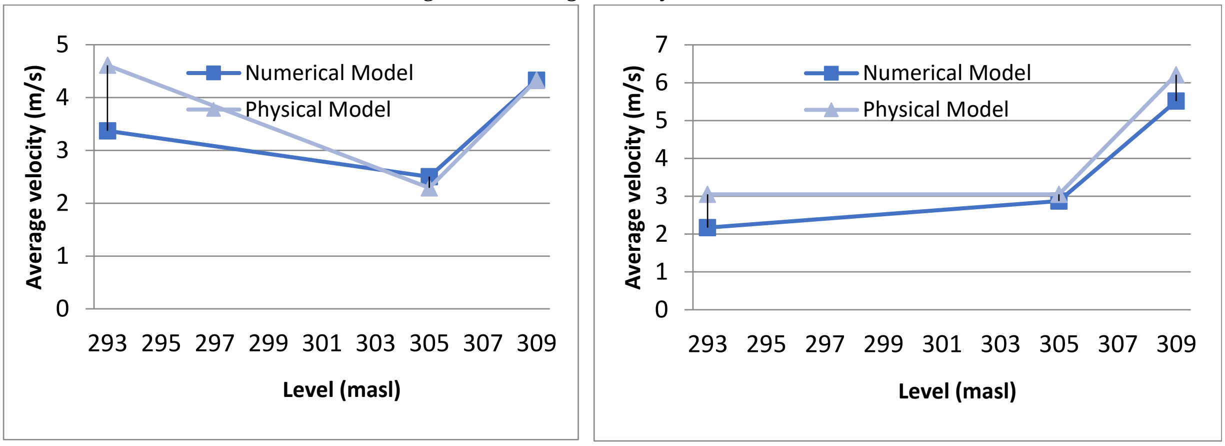


b) point 19

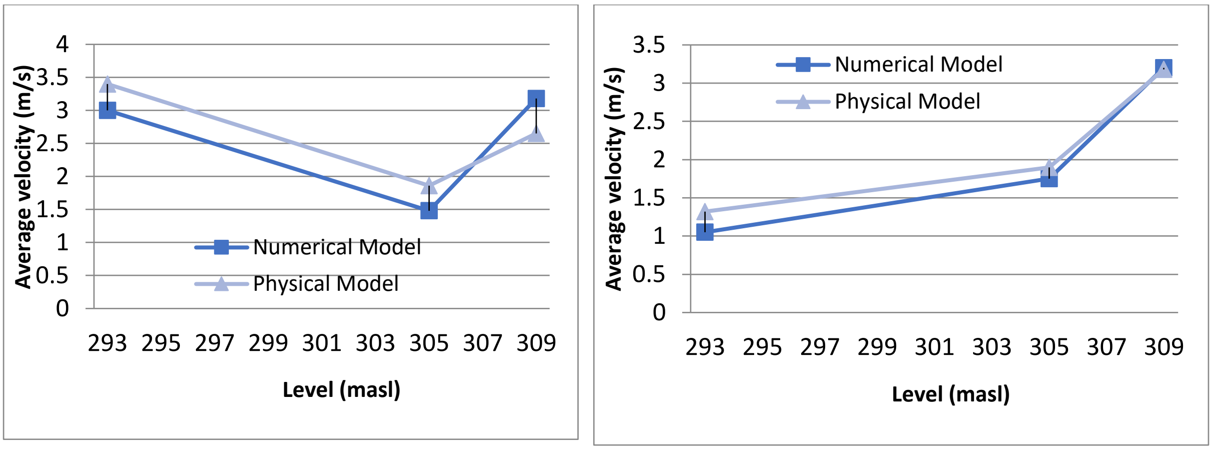


c) point 2 d) point 3

Figure 11. Average velocity for 3000 m^3/s



a) point 9 b) point 19



c) point 2 d) point 3

Figure 12. Average velocity for 4000 m^3/s

After comparing the results, it is found that the maximum error for point 9 is 11.18%, point 19 is 12.02%, point 2 is 29.67%, and for point 3 is 19.24% for 2000 m^3/s flood respectively. Similarly, the maximum errors for 3000 m^3/s flood are 22.16%, 28.50%, 27.47%, and 20.15%, and for 4000 m^3/s flood are 26.86%, 28.96%, 20.56%, and 20.17% for the respective points.

After the validation, the three modification cases: (i) by opening only two spillway gates, (ii) by decreasing the depth of stilling basin and (iii) by shifting the location of auxiliary dam towards upstream are studied for 4000 m^3/s flood.

4.1. Modified case by opening only two spillway gates

A scenario study is conducted by varying the number of openings to test for better results. In this study, a 500-year return period flood (4,000m^3/s) is passed through the two gates by closing one gate. The flow after the crest of the spillway is rapidly varied with very high velocity. The high flow directly strikes the left side of the stilling basin by forming the vertical eddies that can damage the structural parts. Hence, this option is not suggested for the execution.

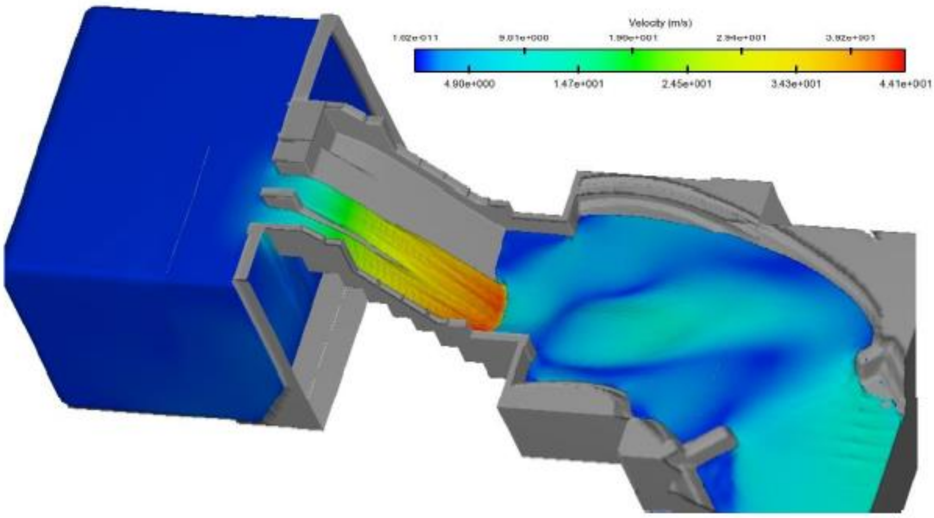


Figure 13. Velocity contour for 4000 m^3/s when two gates are open

4.2. Modified case by decreasing the depth of the stilling basin

Another scenario study is conducted by decreasing the depth of the stilling basin. The depth of stilling basin is filled up to 290 masl, so the volume also be decreased significantly. It is observed that the volume remaining to settle down the turbulent flow inside the stilling basin is reduced. The velocity measured inside the basin was also increased at the different locations, which will enhance the bed scouring and damage the structures. Hence, the reduction of the depth of the stilling basin is not recommended due to the high risk of damaging the hydraulic structures.

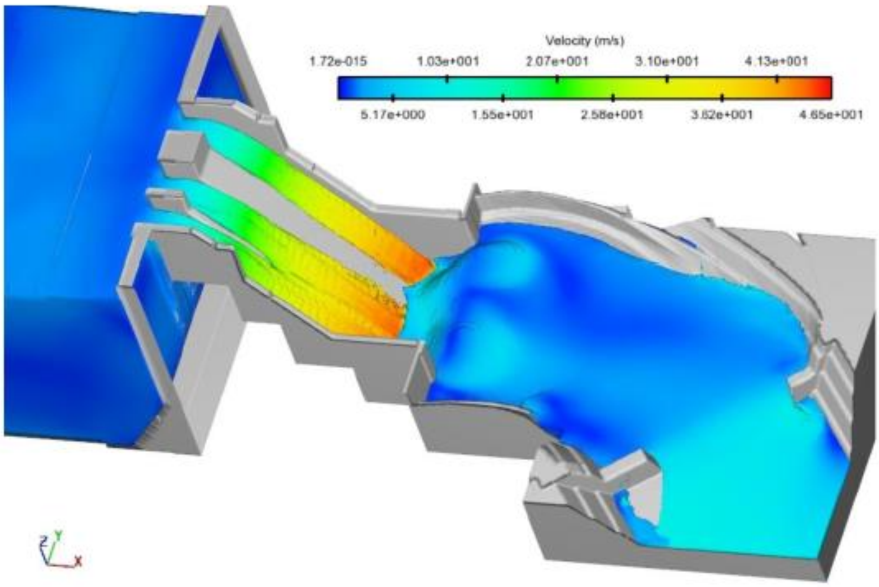


Figure 14. Velocity contour for 4000 m^3/s when the depth of the basin is decreased

4.3. Modified case by shifting the auxiliary dam upstream

Another scenario study is conducted by shifting the location of the auxiliary dam upstream. The length of the stilling basin is reduced while shifting the auxiliary dam by 50 m upstream. It is observed that the length remaining to settle the hydraulic jump is significantly reduced, so it becomes difficult to dissipate the high kinetic energy. The flow velocities inside the stilling basin are also found to be increased which enhances the scouring phenomena. The water surface profile is unstable and continuously fluctuated up to the location of the auxiliary dam. Hence, this case is also not recommended due to being unable to settle to the high kinetic energy by the stilling basin.

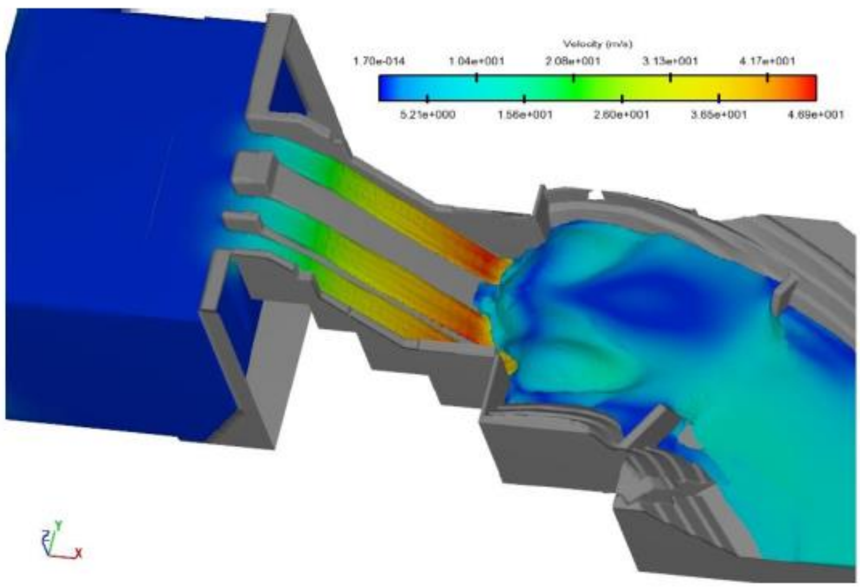


Figure 15. Velocity contour for 4000 m^3/s when the auxiliary dam is shifted upstream

5. Conclusion

Historically, the analysis of flow through the hydraulic structures was studied by scaled physical model in the laboratory. The physical model being uneconomical and facing scaled effect; a numerical technique is attempted for the same purpose in this study. The study is conducted to analyze the flow through the stilling basin by using the Computational Fluid Dynamics technique. Firstly, the model is run for different return period floods to validate the results from the 1:60 scaled physical models. The results of the water surface profile are found within 1.92 m and the flow velocities within the 30% accuracy compared to that of the physical models. The validated model is run for three modified cases to check whether or not the model has an alternative solution. The three scenario studies are conducted by opening only two of the three spillway gates, decreasing the depth of the stilling basin, and decreasing the length of the basin by shifting the auxiliary dam upstream. The results are analyzed, and it is found that none of the three modified case scenarios performed better than the base case scenario without modification. Hence, the base case scenario was the best solution for the effectively dissipation of the high kinetic energy due to the floods. This study recommends that the well setup CFD model is able to precisely predict the experimentally measured quantities. This makes the developed model an effective tool for design and modification processes.

References

Cook, C. B. & Richmond, M.C., 2001. *Simulation of tailrace hydrodynamics using computational fluid dynamics model.* Pacific Northwest National Lab, Richland, WA.

Cook, et.al., 2002. *Free-surface computational fluid dynamics modeling of a spillway and tailrace: Case study of the dalles project* .

Chen, et.al., 2010. *Numerical simulation of the energy dissipation characteristics in stilling basin of multi-horizontal submerged jets.* Journal of Hydrodynamics, 22(5):732-741.

Babaali, et.al., 2015. *Computational modeling of the hydraulic jump in the stilling basin with convergence walls using cfd codes.* Arabian Journal for Science and Engineering, 40(2):381-395.

Bayón, et.al., 2019. *Numerical modeling of multi-horizontal submerged jets (mhsj) stilling basin using cfd techniques.* In 38th IAHR World Congress, Panama City, Panama.

Babaali, et.al., 2019. *Numerical modeling of flow usbr ii stilling basin with end adverse slope.* International Journal of Environmental and Ecological Engineering, 13(2):62-68.

Macián-Pérez, et.al., 2020. *Analysis of the flow in a typified usbr ii stilling basin through a numerical and physical modeing approach.* Water, 12(1):227.

Briggs, M.J., 2013. *Basics of physical modeling in coastal and hydraulic engineering.* ENGINEER RESEARCH AND DEVELOPMENT CENTER VICKSBURG MS COASTAL AND HYDRAULICS LAB..

Thapa, S. & Bogati, P. R., 2012. *Hydraulic model testing of headworks of upper seti hydropower project for detailed engineering study.* Hydro-Lab Pvt. Lt..

Sabbagh-Yazdi, et.al., 2007. *Turbulent modeling effects on finite volume solutionof three dimensional aerated hydraulic jumps using volume of fluid.* 12th WSEAS International Conference on Applied Mathematics, Cairo, Egypt.