Proceedings of 14th IOE Graduate Conference
Peer Reviewed
Year: 2023 Month: December Volume: 14
ISSN: 2350-8914 (Online), 2350-8906 (Print)

Performance Assessment of Hydrocyclone using Numerical Modeling

Shovik Shilpakar [a], Hari Prasad Pandit [b],

[a, b] *Department of Civil Engineering, Pulchowk Campus, IOE, Tribhuvan University, Nepal*
[a] shovikshilpakar@gmail.com, [b] hppioe@gmail.com

Abstract

Every year hydropower plants in Nepal suffer severely due to sediment erosion in turbines and other hydromechanical accessories leading to efficiency losses, unplanned outages and increased maintenance which directly has an adverse impact on revenue generation. Since the conventional gravity settling methods are found inefficient, this research focuses on the application of centrifugal separation for better handling of suspended sediments thorough modeling technique using ANSYS Fluent software. The specific objective of this research is to validate the Numerical model of hydrocyclone with the Physical model and subsequently to analyze the device's performance under different angle of inclination of its axis. The comparison of Numerical and Physical model is made in terms of both water distribution and sediment throughput and a close match between them was achieved. Subsequently, the axis of the device is tilted to 60°, 45° and 30° with respect to horizontal axis from the original vertical axis orientation. The performance of device is then checked and it is found that the headloss in the device decreased by 0.071 m while the separation efficiency is reduced particularly for fine sediments when tilted from 90° to 30°.

Keywords

ANSYS Fluent, AutoCAD, CFD, CFD Post, Headloss, Hydrocyclone

1. Introduction

Nepal is home to around 6,000 rivers with a drainage area of 191,000 sq km., 74 % of which is in Nepal alone [1]. Together with the mountainous terrain of the country, these rivers present enormous prospects for the production of hydropower. However, due to factors like intense monsoon rainfall, young and fragile geology, steep topography and due to influence of South Tibetan Detachment Surface, these rivers carry substantial sediment loads [2]. Consequently, hydropower projects in Nepal face sediment-related challenges, leading to efficiency reduction, unplanned outages and increased maintenance which directly has an adverse impact on the revenue generation from the power production.

In general, Settling Basin are widely used in hydropower plants for excluding suspended sediments coarser than 150-200 microns depending upon the head and generation capacity. However, they are proving insufficient. For instance, In a high head project (920 m), the Pelton turbine is found to have severe erosion followed by cavitation exposed to 77% particles that were finer than 63 microns and 99% that were finer than 125 microns shortly after 600 hours of operation [3]. Settling Basin of Jhimruk HEP, designed to trap particles coarser than 200 microns traps only 17% of total sediment loads due to abundance of fine sediments [2] and, Khimti HEP, 690 m high head project suffers considerable loss due to wear and tear of turbines despite trapping 97% of quartz particles larger than 200 microns and 85% of quartz coarser than 130 mirons [4]. Similarly, the Francis turbines of Trishuli, Panauti and Sunkoshi HEP were frequently eroded and restored by welding and grinding [5].

Since the conventional sediment exclusion methods are found inefficient leading to erosion and efficiency losses in turbines, the exploration of alternatives to conventional settling basin becomes necessary. Therefore, this research focuses on studying the application of Hydrocyclone for improved exclusion of suspended sediments in hydropower projects.

Hydrocyclone is a type of device that is based on the effect of centrifugal force to separate one fluid from another or to separate solid particles from liquid/gas based on their difference in density. It is a simple conical shaped cylindrical vessel without any moving parts in which the required vortex is generated by fluid itself. The schematic sketch of hydrocyclone is presented in Figure 1.

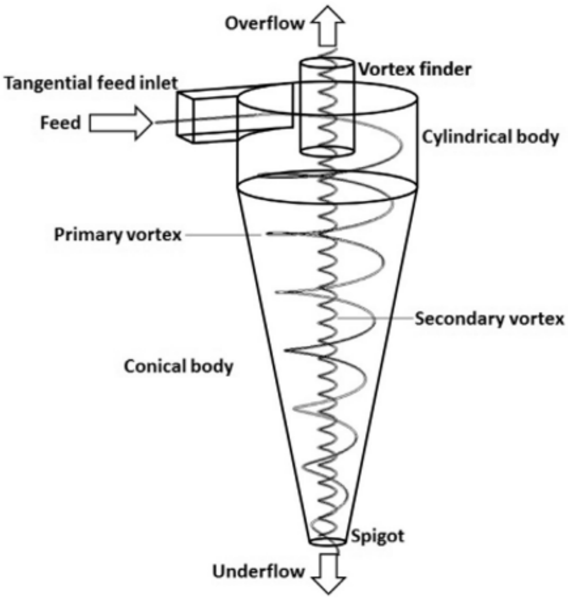


Figure 1: Schematic diagram of hydrocyclone [6]

When fluid with suspended particles enters the hydrocyclone at high velocity through a tangential inlet, it forms a spiral motion. Particles experience centrifugal force pushing them towards the wall and drag force pulling them inward due to the fluid. Heavier particles experience stronger centrifugal force, causing coarser ones to hit the wall earlier and exit via the underflow outlet after losing momentum. The conical section's decreasing radius leads to higher pressure, drawing air from underflow. Swirling air exits through overflow, dragging finer particles and fluid, allowing their escape due to local drag.

2. Objectives

The main objective of this research is to study the application of Hydrocyclone for handling suspended sediments in hydropower projects. The specific objectives include:

1. To compare the performance of Numerical model of Hydrocyclone with the outputs of Physical model.
2. To assess the device's performance under different angle of inclination of its axis.

3. Methodology

This research employs Computational Fluid Dynamics (CFD) through the ANSYS Fluent (Version 2019 R1) software to study the Centrifugal Separation Device. The study is conducted in two stages:

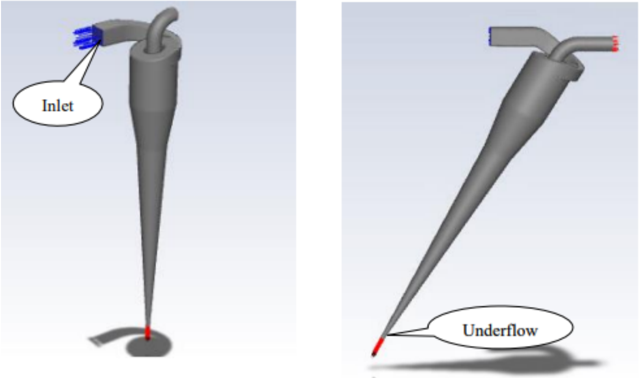
In the initial phase, the Numerical model is validated against a Physical model [2]. Out of 27 total tests runs of Physical model, four of the tests specifically :Test numbers S2-1, S2-3, S2-5, and S2-8 have been taken into as a reference [2]. The primary focus was to verify the hydraulic performance for which the continuity of flow was compared with the results obtained from the Physical model. A detailed Numerical simulation model was developed to replicate the conditions of test number S2-8, which involved the presence of sediments mixed with water.

In the second stage, Keeping Nepal's challenging terrain in mind, this study examines device performance at different inclinations of its axis at 90°, 60°, 45°and 30°with horizontal to reduce overall height potentially easing excavation. In this stage, the sediment excluding efficiency for different particle sizes and headloss in the system is assessed.

The model setup for the ANSYS Fluent involves following steps:

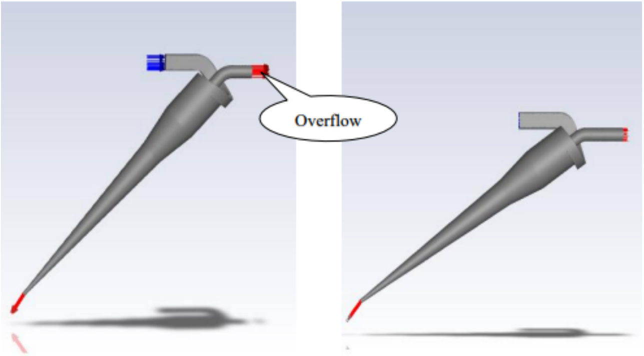
3.1 Geometry Preparation

Creating the test rig geometry is the initial research step, accomplished by preparing a 3D geometry of Hydrocyclone in AutoCAD aligning it with the Physical model's dimensions as shown in Figure 2. The dimensions of test rig are presented in Table 1. For the second phase, the test rig axis was tilted at an angle of 60°, 45° and 30° from its original vertical orientation.



a. Vertical

b. Inclined at 60°



c. Inclined at 45°

d. Inclined at 30°

Figure 2: 3D model of Hydrocyclone for ANSYS Fluent

Table 1: Dimension of test rig of Physical Model [2]

S.N.	Parameter	Unit	Measurement
1	Diameter of hydrocyclone	m	0.38
2	Height of cylindrical part	m	0.50
3	First Cone angle	deg	18
4	Second Cone angle	deg	6
5	Height of first conical part	m	0.4
6	Height of second conical part	m	1.35
7	Variation of underflow aperture	mm	15-60

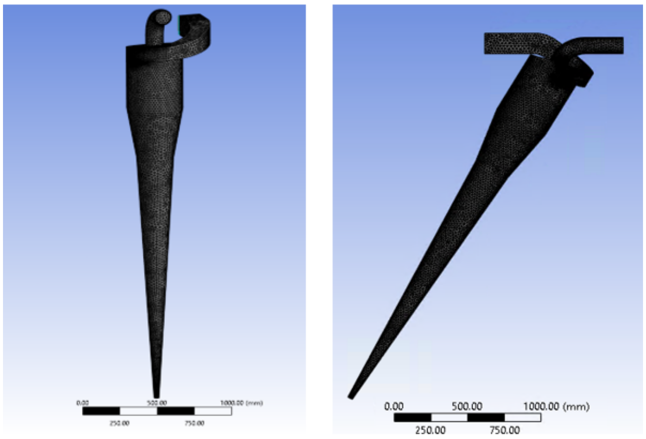
3.2 Meshing

Tetrahedral elements with a global element size of 20 mm are used for the generation of mesh. To capture the boundary layer separation on the walls, three layers of inflation are added with the first layer thickness of 2 $x10^{-3}$ m at a growth rate of 1.2. The number of mesh elements for different orientation of Hydrocyclone is shown in Table 2.

Table 2: Number of elements for different configuration

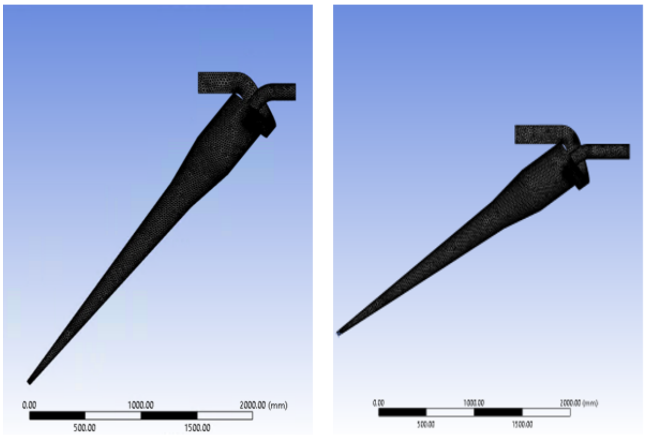
S.N.	Axis Angle with Horizontal	No. of Elements
1	90°	220912
2	60°	284626
3	45°	239314
4	30°	243135

The meshing for the geometry inclined at different angle with horizontal is presented in Figure 3.



a. Vertical

b. Inclined at 60°



c. Inclined at 45°

d. Inclined at 30°

Figure 3: The fully meshed geometry of Hydrocyclone at different angle with horizontal

3.3 Selection of Turbulence and Multiphase Model

To replicate hydrodynamic behavior in the hydrocyclone, selecting the turbulence model is crucial. The swirl modification RNG (k-epsilon) model was chosen for the Numerical simulation of the Hydrocyclone. Despite demanding computational resources, this model yields more precise outcomes in complex flows or near walls [7].

For simulating the Centrifugal Separation Device, the multiphase model was employed, given the water and sediment mixture injection via the inlet. Water served as the primary phase, while sediment was the secondary. Water density and viscosity were set at 998.2 kg/m^3 and 0.001 kg/m/s respectively. Sediment particles density ranged from 2500 kg/m^3 (finer particles) to 2680 kg/m^3 (coarser particles). Sediment size was set within the range of 1 micron to 400 microns.

3.4 Boundary Conditions

In the simulation, the inlet was set as “mass flow inlet”, while the both outlets were defined as “pressure outlets” at atmospheric pressure. Wall boundaries were set to non-slip conditions (velocity set to zero). The mass flow inlet was configured to match the experimental setup of the Physical model. The details of boundary conditions are listed in Table 3.

Table 3: Summary of Boundary Condition

Parameters and Boundary Conditions	**Settings**
Inlet: mass flow rate	16.60 lps – 19.67 lps
Outlet	Pressure Outlet at atmospheric pressure
Walls	No slip for water and partial slip for sediments
Mass flow rate of sediment	0.028 kg/s

3.5 Simulation Run

The simulation commenced by initializing variables such as velocity, pressure, and sediment volume fraction based on system knowledge and experimental data. A total 20,000 iterations were conducted to capture flow turbulence and secure model convergence. The simulation is set to be converged when the difference in residuals between two consecutive iteration is less than 5 x 10^{-3}, confirming a stable and accurate steady state.

3.6 Post Processing

The results were analyzed using CFD-Post and MS Excel. CFD-Post facilitated visualizing flow patterns various contour plots, velocity fields, and streamlines which provided insights into complex fluid behavior.

For comparison between Numerical Model and Physical Model, volume flow rates from both outlets were calculated for different inflow conditions to assess flow continuity and was compared it with Physical model results. Similarly, the mass flow rates of sediments at inlets and both outlets were derived from the model and the results were used to estimate sediment trapping efficiency for different particles size which is calculated using:

$$E = \frac{q_{su}}{q_{su} + q_{so}}$$

Where,
q_{su} = mass flow rate of sediment received from the underflow outlet,
q_{so} = represents the mass flow rate of sediment received from the overflow outlet.

Comparison between Numerical and Physical models was done visually and as well as by using statistical tools like Percentage Bias, RMSE, and R^2.

For estimating headloss within the system, the difference in pressure between inlet and overflow outlet was assessed, expressed in Pascal. The difference in pressure is divided by the specific weight of water to express the headloss in terms of meter.

The process was repeated to estimate sediment separation efficiency for different particles size and headloss within the device at different orientation of device’s axis.

4. Results and Discussions

4.1 Comparison of Numerical Model with and Physical Model

The continuity of flow was checked and the results obtained from simulation is compared with Physical Model [2] and the findings are presented in Table 4.

Table 4: Comparison of flow rate of Physical and Numerical Model in ANSYS Fluent

Test No.	Discharge passing through	Physical Model (l/s)	Numerical Model (l/s)
S2-1	Inlet	17.20	17.20
	Overflow Outlet	14.68	15.35
	Underflow Outlet	2.52	1.84
S2-3	Inlet	19.40	19.40
	Overflow Outlet	17.30	17.33
	Underflow Outlet	2.10	2.06
S2-5	Inlet	17.85	17.85
	Overflow Outlet	15.45	15.97
	Underflow Outlet	2.40	1.87
S2-8	Inlet	16.60	16.60
	Overflow Outlet	14.33	14.52
	Underflow Outlet	2.27	2.07

Figure 4 displays the post-processed simulation outcome depicting water velocity vector. The velocity streamline depicts that the maximum velocity of flow can be observed near the inlet to the Hydrocyclone where the flow is accelerated due to reduction in the cross-sectional area from 0.15 m x 0.15 m squared section to 0.055 m x 0.11 m rectangular section. A gradual decrease in velocity towards the bottom of the device is observed. There is a flow directing downward in the outer circumferential region responsible for directing the sediments towards underflow outlet whereas a relatively low velocity zone in the inner core of the device is directed in upward direction, which is responsible for directing the water towards overflow outlet as demonstrated in Figure 4.

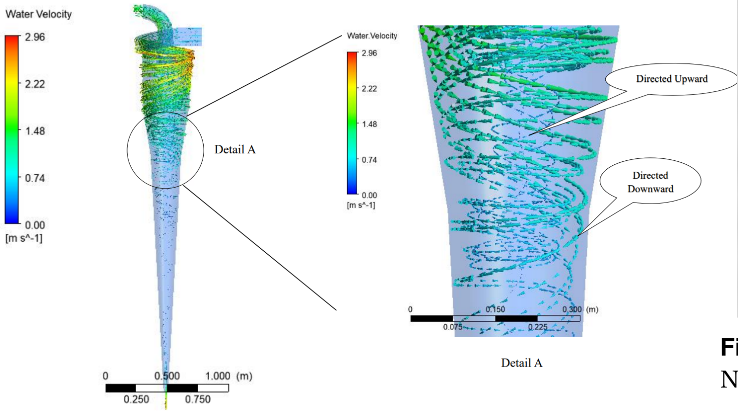


Figure 4: Velocity vector depicting the flow direction inside hydrocyclone at Q_{in} = 16.60 lps

The sediment trapping efficiencies for particles of different sizes were evaluated, and the comparison of the simulation results with output of Physical model is illustrated in Figure 5.

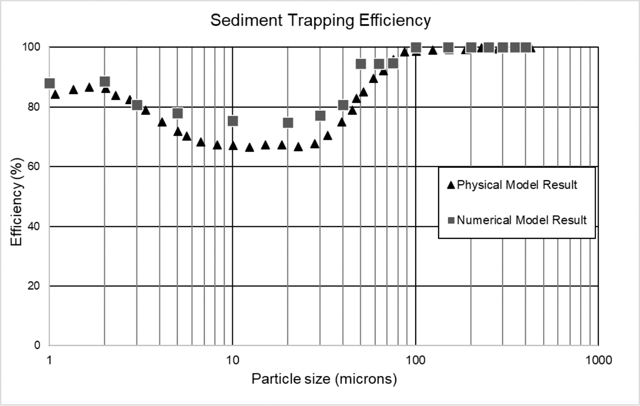


Figure 5: Comparison of sediment trapping efficiency for different size of sediments

A fish hook effect can be observed in Figure 5. This effect is a characteristic inherent to hydrocyclone, where a slight increase in efficiency is observed at a very fine particle sizes before decreasing and again increasing for coarser particle sizes [2]. The observation of this fish hook effect provides further validation of its capability to accurately capture the distinct properties and behavior of the hydrocyclone device.

Visual comparison reveals close alignment of sediment trapping efficiency between Numerical and Physical models, with slight enhancement in Numerical efficiency. Quantitative assessment includes statistical parameters: PBIAS as -3.40, RMSE as 4.407, and R^2 as 0.96, indicating strong resemblance. Efficiencies for different sediment sizes, shown in Figure 5, maintain consistency between Numerical and Physical models, affirming Numerical simulation's reliability in accurately depicting sediment trapping behavior. Similarly, the graph illustrating efficiencies for different sediment sizes from both Physical and Numerical Model is presented in Figure 6.

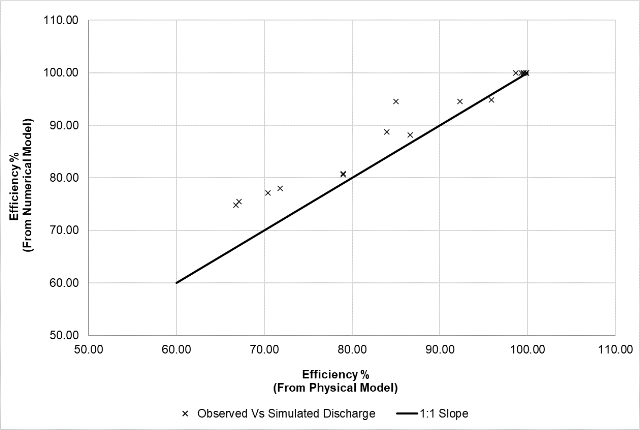


Figure 6: Comparison of trap efficiency of Physical and Numerical Model

Similarly, the trajectories of sediment particles were tracked simultaneously. The streamlines of sediments particles illustrating their velocity distribution is presented in Figure 7. Clearly it can be observed that, of all of the sediment particles that is fed upon the Hydrocyclone, majority of the sediment particles are directed towards the underflow outlet while only a few escapes from the overflow outlet showing the highly efficient character of Hydrocyclone in separating sediments from mixture of sediments and water.

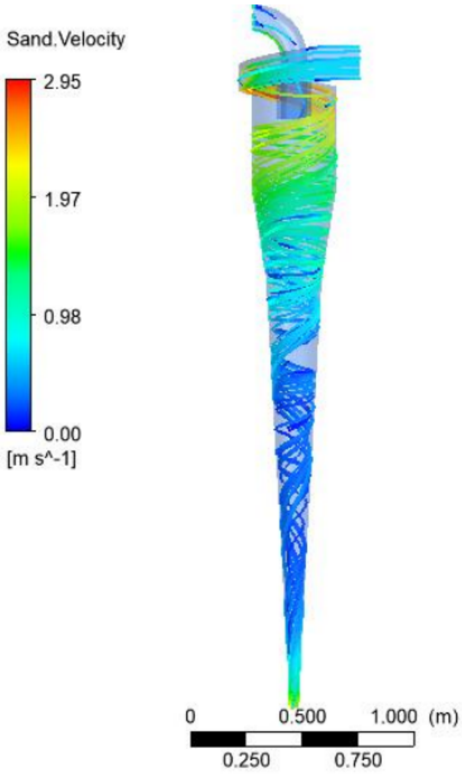


Figure 7: Velocity streamline of sediments in the hydrocyclone at Q_{in} = 16.60 lps

4.2 Performance of Hydrocyclone at Different Inclination of its Axis

The sediment trapping efficiency for various sizes ranging from 1 micron to 400 micron for different inclination of device's axis with horizontal are presented in Figure 8, Figure 9, Figure 10 and Figure 11 respectively.

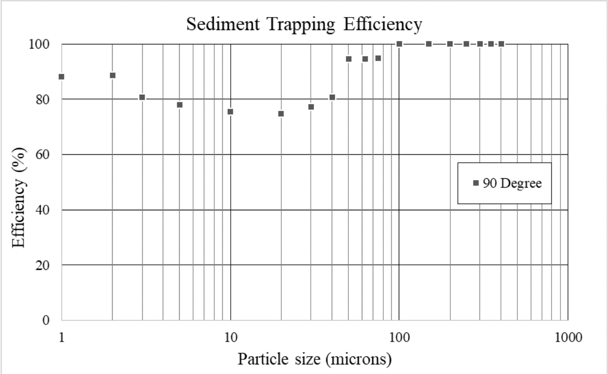


Figure 8: Sediment trapping efficiency for different size of sediments for vertical orientation

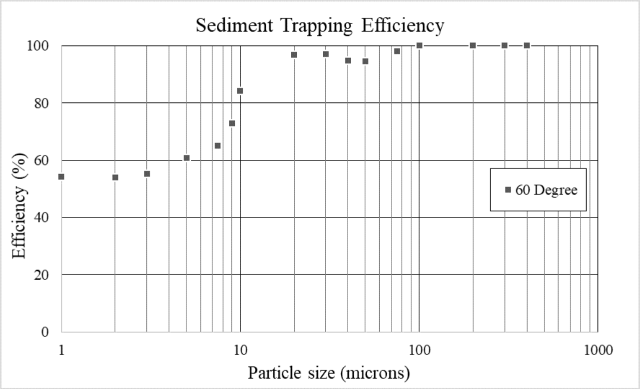


Figure 9: Sediment trapping efficiency for different size of sediments for 60° inclined hydrocyclone

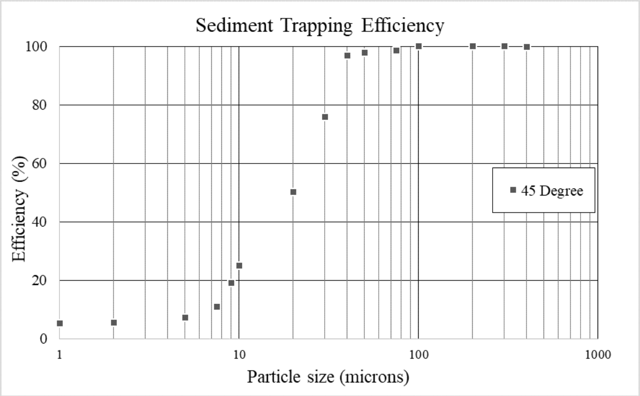


Figure 10: Sediment trapping efficiency for different size of sediments for 45° inclined hydrocyclone

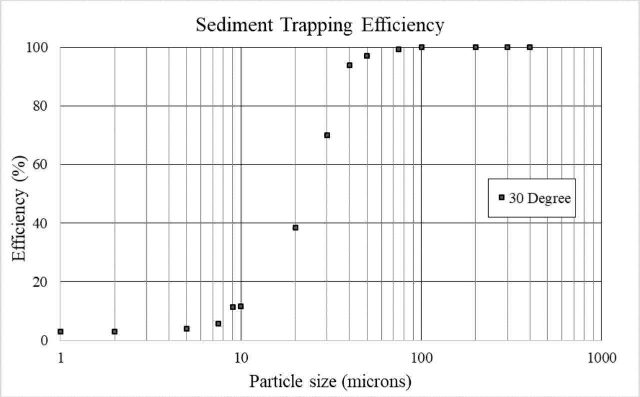


Figure 11: Sediment trapping efficiency for different size of sediments for 30° inclined hydrocyclone

A figure depicting the comparison of sediment trapping efficiency for different size of sediments for different inclination of device is summarized in Figure 12.

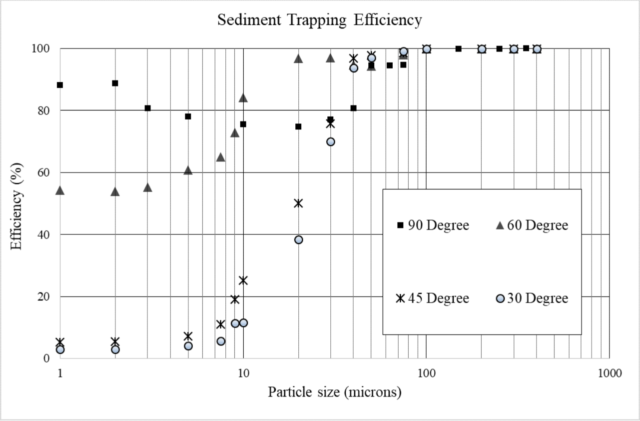


Figure 12: Comparison of sediment trapping efficiency for different size of sediments for different inclination of hydrocyclone

Similarly, the headloss in the device was studied at different configuration of its's axis inclined at 90°, 60°, 45° and 30° to horizontal respectively which is presented in Table 5.

Table 5: Headloss in Hydrocyclone at different orientation for Q_{in} = 16.60 lps

S.N.	Axis Angle with Horizontal	Headloss (m)
1	90°	0.758
2	60°	0.709
3	45°	0.695
4	30°	0.687

The headloss in the device depends on flow rate, size and shape of hydrocyclone, density of fluid and concentration of solids in the fluid. Table 5 shows that even for the small discharge, the head loss is very high. The high headloss is caused due to the high turbulence flow, induced by centrifugal force and complex hydraulics inside it. By reducing the angle of inclination from 90° to 30°, the headloss within the system is reduced by 0.071 m. This reduction in headloss is desirable from the generational point of view, however it can be observed in Figure 8, Figure 9, Figure 10 and Figure 11 that while tilting the hydrocyclone from 90° , to 30° , the separation efficiency is reduced particularly for fine sediments i.e. less than 75 microns [8] while the coarser sediments i.e. greater than 75 microns [8], are almost removed.

5. Conclusion

This study conducted a thorough analysis of Hydrocyclone for sediment-water separation in hydropower projects. First, the Numerical model was validated with the Physical model and subsequently the performance of device was assessed after tilting the device's axis to different angle with respect to horizontal axis to reduce the overall height of device easing the excavation issue considering varied topography of Nepal.

The key conclusions from this study are as follows:

1. The Centrifugal Separation Device's CFD simulation closely matched the Physical Model, displaying similar water distribution and sediment throughput. Statistical tool was assessed to compare the sediment separation efficiency of both Physical and Numerical Models and the value of PBIAS, RMSE and R^2 were observed to be -3.40, 4.407 and 0.96. The alignment of Numerical and Physical models in hydraulic and sediment trapping behaviors demonstrated the Numerical simulation's dependability and correctness. Similarly, the observation of fish hook effect provided further validation of the model to accurately capture the inherent character and behavior of Hydrocyclone. This served as a robust foundation for ensuring accuracy and dependability in subsequent study stages.
2. The device's performance was evaluated at varying angles of inclination. Initially vertical, the Hydrocyclone's axis was adjusted to incline at 60°, 45°, and 30° relative to the horizontal. As the inclination decreased from 90° to 30°, the headloss in the system decreased by 0.071 m. While this headloss reduction is beneficial, it coincided with reduced efficiency in separating sediments, particularly finer particles. Therefore, it is a trade-off between the head and separation efficiency, so it is important to find the balance between the sediment separation efficiency and headloss to meet the specific objective of the application.

Acknowledgments

The authors would like to thank NEA Engineering Company for providing access to ANSYS Fluent software for the research. The authors would also like to thank Er Smarika Tamrakar and Er Sulav Parajuli for their guidance during the research.

References

[1] Bikram Manandhar. Flood plain analysis and risk assessment of lothar khola. *Master of Science Thesis in Watershed Management. Tribhuvan University Institute of Forestry Pokhara, Nepal*, 2010.

[2] HP Pandit, NM Shakya, H Stole, and NK Garg. Sediment exclusion in himalayan rivers using hydrocyclone. *ISH Journal of Hydraulic Engineering*, 14(2):118–133, 2008.

[3] H Brekke, YL Wu, and BY Cai. *Design of hydraulic machinery working in sand laden water*, volume 2. Imperial College Press London, 2002.

[4] IM Deshar. O and m experience in sediment management. Workshop on Operations, Repair and Maintenance of Hydropower Plants, Kathmandu, January 2007. BPC and Statkraft Norfund Power Invest AS.

[5] Roman Gantawa and Naresh Kazi Tamrakar. Analysis of sediment abrasion potential in hydro turbines by studying fine sediments from the budhi gandaki-trishuli river of nepal. *Journal of Nepal Geological Society*, 58:29–38, 2019.

[6] D Vega-Garcia, PR Brito-Parada, and JJ Cilliers. Optimising small hydrocyclone design using 3d printing and cfd simulations. *Chemical Engineering Journal*, 350:653–659, 2018.

[7] Victor Yakhot and Steven A Orszag. Renormalization group analysis of turbulence. i. basic theory. *Journal of scientific computing*, 1(1):3–51, 1986.

[8] TW Lambe and RV Whitman. Soil mechanics. new york: Johnwiley and sons, 1969.