Draft Tube Design and Analysis

Aashish Katuwal, Bipin Shrestha, Himal Timsina, Upendra Gautam

Tribhuvan University, Institute of Engineering

August 10, 2023

Abstract

This paper aims to report the finding from our ANSYS simulation on flow through a draft tube where multiple types of draft tube were analysed via finite element method. Flow visualization, velocity contour, pressure contour are analyzed and the results tabulated. The resulting simulation values are then compared with the theoretical values obtained using equations at ideal conditions. The efficiency calculated are theoretical. The report aims to illustrate the significance of geometry towards efficiency While the principal intention and purpose of this paper has been to analyze the results through thorough explanations, commitment to latter dwindled with progress. Despite some shortcomings, there's always a realization that certain data associated with this paper could be of value for further research.

1 Introduction

Hydropower is the primary source of energy in Nepal, contributing significantly to the country's energy generation and economic development. With its abundant water resources and mountainous terrain, Nepal has immense hydropower potential. Hydropower plays a pivotal role in meeting the energy demands of the nation, powering industries, households, and supporting economic growth. As of the latest available data, Nepal generates approximately 2,532.36 megawatts (MW) of electricity from hydropower, contributing around 83% of the country's total energy generation.[5] However, the energy demand continues to grow, driven by increasing urbanization and industrialization. Nepal's annual electricity consumption is estimated to be around 7000 gigawatt-hours (GWh), and the demand is projected to rise further.[4] To bridge the gap between supply and demand, Nepal imports electricity from neighboring countries, primarily India. Despite hydropower's dominance, around 17% of the energy needs are fulfilled through imports, underlining the importance of harnessing the country's hydropower potential efficiently. Efficiency in hydropower is crucial for energy savings and sustainability. Higher efficiency translates to greater electricity output from the same amount of water, minimizing wastage and maximizing utilization of resources. Improved efficiency also reduces environmental impacts and operational costs. However, hydropower development in Nepal is not without its challenges. Geological complexities, varying flow rates, and seismic risks pose engineering challenges. Sediment deposition in reservoirs, which diminishes storage capacity and affects turbine efficiency, is a significant concern. To address these issues, it's essential to come up with innovative solutions such as advanced sediment management strategies, innovative turbine designs to handle variable flow conditions, and earthquake-resistant structures to ensure the safety and reliability of hydropower installations. Each component of the hydropower plant contribute to overall efficiency of the plant. For a highly efficient plant, it's crucial to have each and every component of the hydropowerplant be optimized according to external factors. Draft tube contributes significantly to the efficiency of a hydropower turbine and overall efficiency. A well-designed draft tube can have a substantial impact on the perforance of the turbine and the amount of energy extracted from the flowing water. Draft-tube is responsible for energy recovery, reduced htdraulic loss, cavitation prevention, operational stability.

The draft tube is a conduit which connects the runner exit to the tail race where the water is being finally discharged from the turbine. The primary function of the draft tube is to reduce the velocity of the discharged water to minimize the loss of kinetic energy at the outlet. This permits the turbine to be set above the tail water without any appreciable drop of available head[6]. Additional advantages of the draft tube is, it enables placing the runner above the tail water without losing head, as well as directs the water flow into the tail water.[3] The turbine's exit is equipped with a draft tube that boosts the fluid's pressure while sacrificing its speed. Consequently, the turbine can effectively decrease pressure to a greater degree without concerns about reverse flow from the tail race.[7]

2 Draft Tube of Francis Turbine

In the scenario of impulse turbines, where there's a substantial vertical drop in water level, elevating the turbine a short distance above the tail race doesn't notably impact its efficiency. However, for reaction turbines operating with a lower net head, placing the turbine above the tail race could lead to a notable reduction in available pressure energy for powering the turbine. Furthermore, if the fluid pressure in the tail race surpasses that at the turbine exit, it could result in harmful backflow of liquid into the turbine, causing significant damage.[7]
Type of draft tube:
1. Conical diffuser or straight divergent tube: This type of draft tube consists of a conical diffuser with half angle generally less than equal to 10° to prevent flow separation. It is usually employed for low specific speed, vertical shaft francis turbine. Efficiency of this type of draft tube is 90%
2. Simple elbow type draft Tube: It consists of an extended elbow type tube. Generally, used when turbine has to be placed close to the tailrace. It helps to cut down the cost of excavation and the exit diameter should be as large as possible to recover kinetic energy at the outlet of runner. Efficiency of this kind of draft tube is less almost 60%
3. Elbow with varying cross section – It is similar to the Bent Draft tube except the bent part is of varying cross section with rectangular outlet.the horizontal portion of draft tube is generally inclined upwards to prevent entry of air from the exit end.[7]

3 Methodology Adopted

3.1 Geometric Model

The following systematic diagram shows the geometry of the draft tube used for simulation while the simple conical type draft tube has inlet diameter 26.157 mm, divergence angle of 5° and lenth 350 mm.

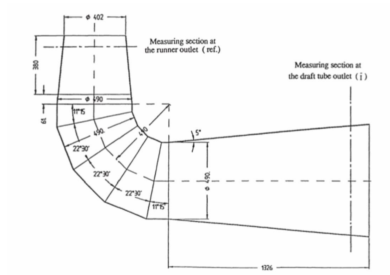

Figure 1: Geometry for elbow type draft tube

3.2 Meshing

Descritization of geometric model is done and terahedral mesh has been generated using ICEM CFD Software Package.[1] The following are the mesh conditions used:
a. Simple Conical

Element size: 5mm
Meshing method: Tetrahedron

Domain	Nodes	Elements
fff_solid	13085	67910

Figure 2: Mesh report for Simple Conical type draft tube

b. Conical Elbow and Rectangular Elbow

Element Size: 40mm
Mashing method: Tetrahedron

Domain	Nodes	Elements
fff_solid	14949	79366

Figure 3: Mesh report for Concial Elbow type draft tube

Domain	Nodes	Elements
fff_solid	17009	85895

Figure 4: Mesh report for Rectangular Elbow type draft tube

3.3 Mid-plane velocity contour

Velocity contours in ANSYS show how fast fluid is moving in different parts of a computer simulation. These contours use colors to tell us how fast the fluid is moving – different colors mean different speeds. This provides visual context regarding how fluid flows, find places where it's moving fast or slow, and make things work better, like making cars more aerodynamic or understanding how liquids heat up.[2] The figure below shows the velocity contour for different types of draft tube design.

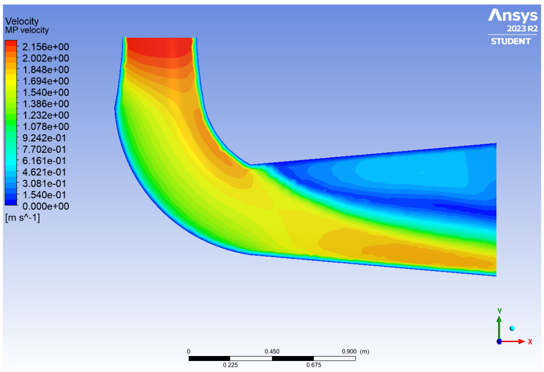

Figure 5: Mid-plane velocity contour for Conical Elbow draft tube

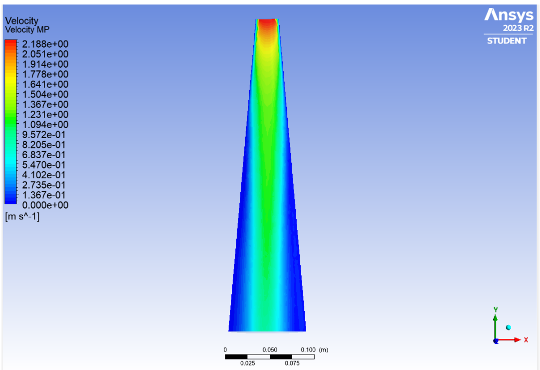

Figure 6: Mid-plane velocity contour for Simple Conical type draft tube

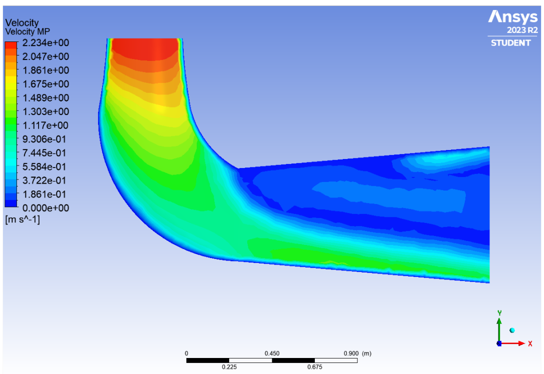

Figure 7: Mid-Plane velocity contour for Elbow draft tube with varying cross-section

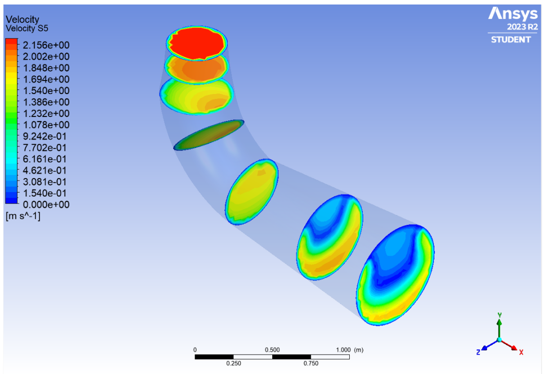

Figure 8: Velocity contour for Conical Elbow draft tube

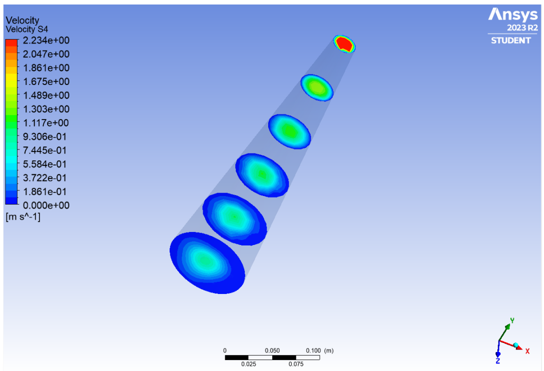

Figure 9: Velocity contour for Simple Conical type draft tube

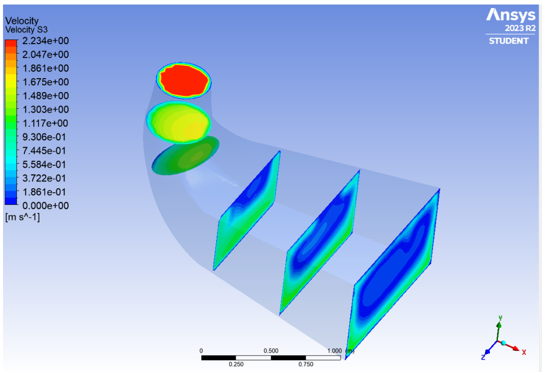

Figure 10: Velocity contour for Elbow draft tube with varying cross-section

3.4 Mid-plane pressure contour

Pressure contours show how pressure spreads out in a simulation.They refer to visual representations that display the distribution of pressure values across a simulation domain. Pressure contours use color gradients to illustrate varying pressure magnitudes, helping engineers and researchers comprehend pressure distribution and variations within structures or fluid domains. These contour plots provide valuable insights into how pressures change across surfaces, aiding in the optimization of designs, identifying potential weak points, and ensuring the structural integrity of components. These contours colors show different pressures, showcasing where pressure is high or low. The figure below shows the preesure contour for different types of draft tube.

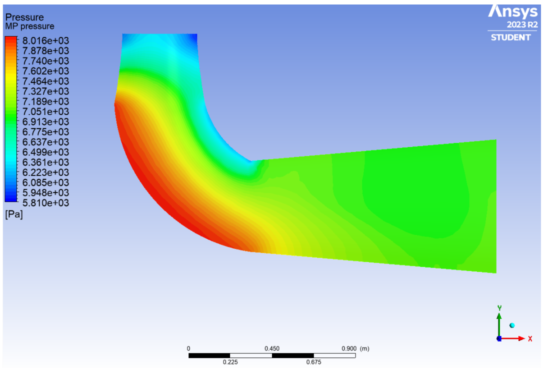

Figure 11: Mid-plane pressure contour for Conical Elbow draft tube

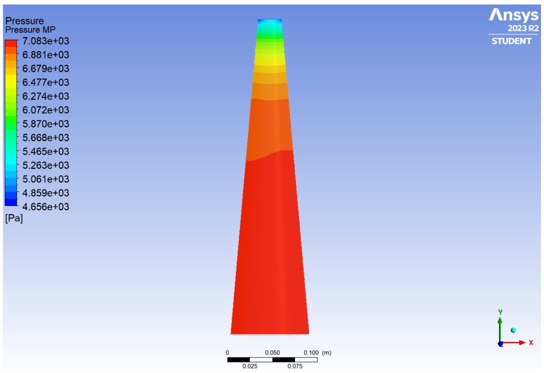

Figure 12: Mid-plane pressure contour for Simple Conical type draft tube

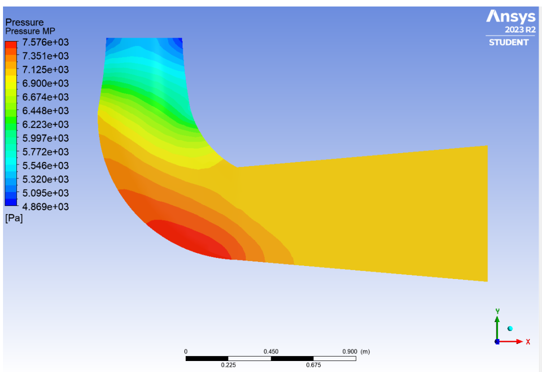

Figure 13: Pressure contour for Elbow draft tube with varying cross-section

3.5 Streamline

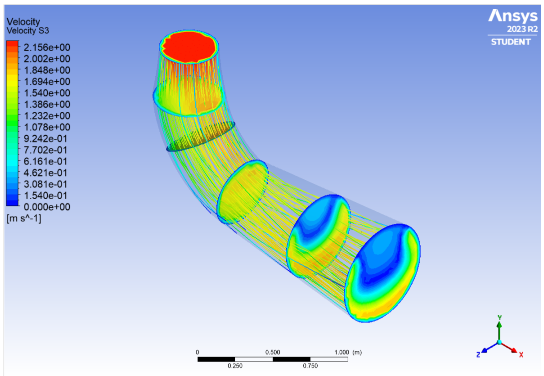

Figure 14: Streamline in a Conical Elbow draft tube

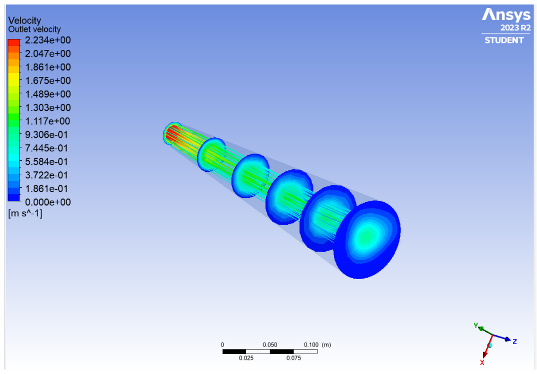

Figure 15: Streamline in a Simple Conical type draft tube

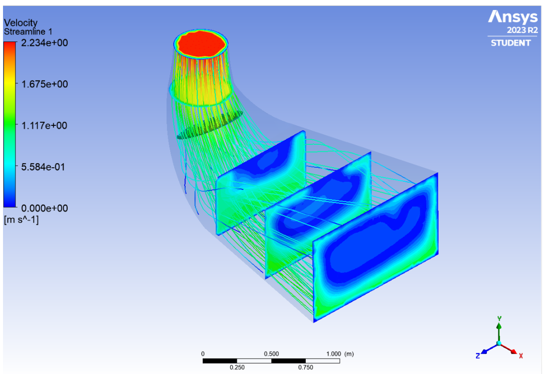

Figure 16: Streamline for Elbow draft tube with varying cross-section

A streamline represents the path that a fluid or particle takes within a flow field. It's a line that follows the direction of the flow at each point, showing the journey that the fluid or particle would follow. Streamlines helps to visualize the flow patterns, identify areas of turbulence or stagnation, and understand how fluids move around objects or obstacles. This is important for designing efficient and effective systems, like optimizing the shape of objects to reduce resistance or improving the efficiency of fluid transportation.

3.6 Equations Involved and Efficiency Calculation

inlet diameter(d_1) = 490 mm
flow rate(Q) = 0.12 m^3/s

Bernoulli's Equation is given as:

$$\frac{P_1}{g*\rho}+\frac{v_1^2}{2*g}+Z_1=\frac{P_{atm}+P_{gauge}}{g*\rho}+\frac{v_2^2}{2*g}+Z_2+h_f \quad (1)$$

(a) Conical Elbow

Outlet diameter (d_2) = 722 mm
Theoretical Outlet velocity (v_2) =

$$\frac{Q}{\frac{\pi}{4}*d_2^2}$$

= 0.2931 m/s
CFD resulting velocity (v_{cfd}) = 0.8704 m/s

Using $eq^n(1)$, we get

Suction Pressure = 94478.8032 Pa

(b) Variable Area Elbow

Outlet Area(A) = 923765.84 mm^2
Theoretical Outlet velocity (v_2) =

$$\frac{Q}{A*d_2^2}$$

= 0.1299 m/s
CFD resulting velocity (v_{cfd}) = 0.3966 m/s

Using $eq^n(1)$, we get

Suction Pressure = 94169.6064 Pa

(c) Simple Conical

Outlet diameter(d_2) = 722 mm
Theoretical Outlet velocity (v_2) =

$$\frac{Q}{\frac{\pi}{4}*d_2^2}$$

= 0.2931 m/s
CFD resulting velocity (v_{cfd}) = 0.2037 m/s

Using $eq^n(1)$, we get

Suction Pressure = 94273.6450 Pa

Efficiency(η) of a draft tube is given by[8];

$$\eta=\frac{\frac{v_1^2-v_2^2}{2*g}-h_f}{\frac{v_1^2}{2*g}}*100 \quad (2)$$

Now,
Efficiency of Conical Elbow (η) = 97.49%

Efficiency of Variable Area Elbow (η) = 98.87%

Efficiency of Simple Conical (η) = 98.38%

3.7 Recovery Head

Recovery Head efficiency for the draft tube is gives as[1]:

$$\eta=\frac{\frac{v_1^2-v_2^2-0.25*v_2^2}{2*g}}{\frac{v_1^2-v_2^2}{2*g}} \quad (3)$$

Now,
Efficiency of Conical Elbow (η) = 95.52%

Efficiency of Variable Area Elbow (η) = 99.16%

Efficiency of Simple Conical (η) = 99.79%

4 Conclusion

From above calculations it's seen that the most efficient design for draft tube is that of Variable Area Elbow draft tube followed by Simple Conical and then Conical Elbow type draft tube. We can conclude that the shape and geometry of the draft tube play a significant role in determining its efficiency. Specifically, factors such as the expansion angle, length, and curvature of the draft tube influence how effectively it converts kinetic energy into pressure energy. Additionally, the flow conditions, including the inlet velocity and discharge, impact the draft tube's performance.

5 Bibliography

References

[1] Harish H V Santhosh N Srikanth H V Dr M S Raghu Praveena B A Amaresha Patil Shivashankar Palakshaiah. "Optimum Design of Conical Draft Tube by Analysis of Flow Using CFD Simulation". In: (2014). URL: https://www.researchgate.net/publication/354270128_Optimum_Design_of_Conical_Draft_Tube_by_Analysis_of_Flow_Using_CFD_Simulation.

[2] "ANSYS velocity contour". In: (2020). URL: https://forum.ansys.com/forums/topic/how-to-get-mean-velocity-contours-in-ansys-fluent/.

[3] I Herland. "Design optimization of Francis turbine draft tube". In: (2020). URL: https://ttl.ku.edu.np/wp-content/uploads/2020/08/CRHTX-05-Ingelin.pdf.

[4] "Nepal Electricity Consumption: Total". In: (2022). URL: https://www.ceicdata.com/en/nepal/electricity-consumption/electricity-consumption-total.

[5] Republica. "Nepal's electricity production reaches 2,689.83 MW; capacity to cross 2,800 MW by this FY end". In: (2023). URL: https://myrepublica.nagariknetwork.com/news/nepal-s-electricity-production-becomes-2-689-83-mw-capacity-to-cross-2-800-mw-by-this-fy-end/.

[6] *Draft Tube*. URL: https://web.archive.org/web/20130510200603/http://nptel.iitm.ac.in/courses/Webcourse-contents/IIT-KANPUR/machine/chapter_7/7_7.html.

[7] "Purpose of draft-tube". In: (). URL: https://en.wikipedia.org/wiki/Draft_tube.

[8] Abhas Chincholikar Vishnu Prasad Ruchi Khare. "Numerical Simulation for Performance of Elbow Draft Tube at Different Geometric Configurations". In: (NA). URL: https://www.researchgate.net/publication/265982012_Numerical_Simulation_for_Performance_of_Elbow_Draft_Tube_at_Different_Geometric_Configurations.