International Research Journal of Advanced Engineering and Science
ISSN (Online): 2455-9024

A Review Paper on Design of Elbow Draft Tube for Unsteady Flow

Jitendra Gupta[1], Santosh Sahu[2]
[1, 2]Mechanical Engg. Dept., Rungta College of Engineering and Technology, Raipur, Chhattisgarh, India-492001

Abstract— *In this research our main objective is to design a elbow draft tube with varying diffuser angle and to find out the optimized model for the same or with enhanced diffuser angle. In order to enhance the design methodology as compare to the conventional draft tube design, we are adopting the method to resolve the problem of draft tube design and to reduce the cost of prototyping and to get the improved results by using computational fluid analysis in ANSYS 14.0 by CFX solver*.

Keywords— *CFD analysis, draft tube, pressure, stream line*.

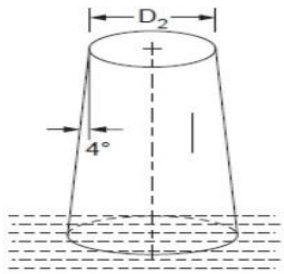
I. INTRODUCTION

The draft tube is one of the important components of Hydraulic reaction turbine and it connects the runner exit to the tail race. The main functions of draft tube is to allow the installation of turbine above the tail race level without loss of head and to convert major part of kinetic energy at the runner outlet into pressure energy . With the use of very low head and high speed turbines, the kinetic energy leaving the runner became higher and the height of the runner above the tail race became smaller. This is achieved by increasing the cross-sectional area of the draft tube in the flow direction. Initially, the straight conical tubes with inlet and outlet areas of different cross-section were used and part of kinetic energy could be converted into potential energy for use in the turbine.

The hydrodynamic investigations on straight draft tubes were carried out between 1903 and 1907 and investigators adopted walls of draft tube parallel to streamline based on theoretical solutions. However, it did not find much practical application due to non-uniform distribution of flow at the entry, swirling of flow and vortex flow along the length of the tube. A large number of investigations were carried out on straight diffusers during in the period 1909 to 1929 lead the design of various draft tubes (reference). The use of straight tubes was restricted to turbines of medium and small diameters because with the increase of the diameter of the runner, the length of the tube became so large that became irrational to construct such tubes. The recovery of the kinetic energy of axial and rotational flow can be best affected in bell mouth tubes. The use of such tubes for large runner diameters has again restriction due to support problem of such large dimensions and weight. All these problems are overcome by elbow draft tube for large diameter hydraulic turbines. The determination of optimum shape and dimension of the draft tube is a very difficult problem and has not been solved till now. The height of curved draft tube has a great influence on the efficiency and power output of turbine.

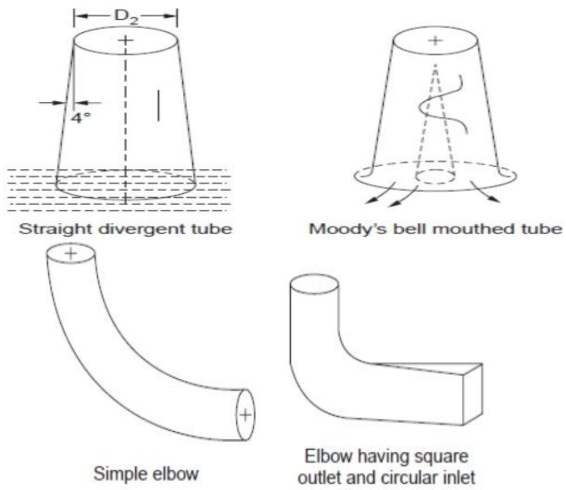
A. Properties of Draft Tube

- Draft tube is a divergent tube one end is connected to the outlet of the turbine and other end is immersed well below the water level.
- The purpose of the draft tube of a water turbine is to reduce the exit velocity with a minimum loss of energy.
- The draft tube ‘converts’ the kinetic energy into pressure energy.



- Geometrically the draft tube is a fairly simple device, a bending pipe diverging in the stream wise direction.
- Dynamical processes of the flow in a draft tube is very complex and many unsteady effects have been observed.
- Draft tubes can be designed in slightly different ways, but some design variables are of less importance than others.
- The shape of the outlet, circular or rectangular, is often of less importance than the outlet area.

B. Types of Draft Tube



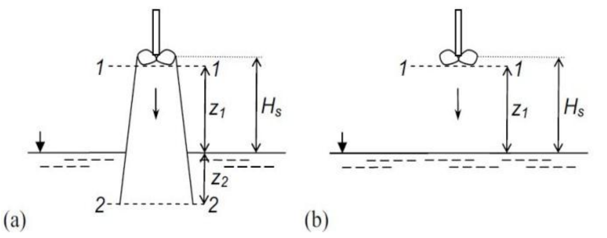
C. Need to Work on Draft Tube

- The draft tube plays an important role on overall performance of reaction turbine.
- The efficiency of draft tube depends on its shape and other geometric parameter the most commonly used draft tube is elbow draft tube with rectangular outlet.
- The shape and area of draft tube at outlet as well as along its length may improve the efficiency of turbine.

Jitendra Gupta and Santosh Sahu, “A review paper on design of elbow draft tube for unsteady flow,” *International Research Journal of Advanced Engineering and Science*, Volume 2, Issue 1, pp. 144-147, 2017.

D. Principles of Draft Tube

The principle of draft tube is determined by the use of Bernoulli's Equation between section inlet 1-1 and outlet 2-2 in fig.-



$$\frac{p_1}{\rho g}+z_1+\frac{V_1^2}{2g}=\frac{p_2}{\rho g}+\frac{V_2^2}{2g}+h_f$$

- The absolute pressure at section 2-2 can also be defined as

$$\frac{p_2}{\rho g}=Z_2+\frac{p_{atm}}{\rho g}$$

- hence equation becomes,

$$\frac{p_1}{\rho g}=\frac{p_{atm}}{\rho g}-\left(H_S+\left(\frac{V_1^2}{2g}-\frac{V_2^2}{2g}-h_f\right)\right)$$

II. Previous Work

The efficiency of a hydraulic reaction turbine is significantly affected by the performance of its draft tube. The shape and velocity distribution at the inlet are, in next turn, two main factors that affects the performance of the draft tube. Traditionally, the design of this component has been based on simplified analytic methods, experimental rules of thumb and model tests. An attempt has been made for design automation of modeling of draft tube using Excel spreadsheet and Creo parametric software. In the last decade or two, the usage of computational fluid dynamics (CFD) has dramatically increased in the design process and will continue to grow due to is flexibility and cost-effectiveness. A CFD-based design search can further be aided with a robust and user-friendly optimization frame work theory and engineering. In this paper, the CFD analysis of draft tube has been performed and results for the same are compared with experimental reading and which are found within the limit [1].

Design Automation of Draft Tube

As per the client's specification, dimensions for draft tube have been decided and 2D drawing for the same has been prepared as shown in Fig. 1.

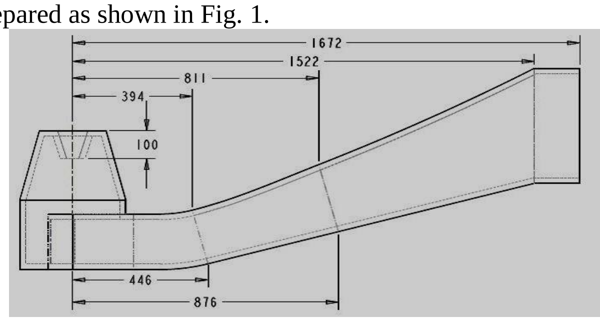


Fig. 1. 2D drawing of draft tube.

Pro/Program is a powerful secondary generation tool to validate parametric design of the component. Pro/Program reflects all parameters and geometric data of the part in a text data form. This data can be modified to new feature, deleting existing feature, suppress the feature and change the dimension of the feature. A proper user interface (API) can directly modify the Pro/Program and the part modal can be driven according to the user input in user interfacing. The boundary condition for draft tube analysis have been applied. In boundary condition inlet mass flow rate is given 1000 kg/sec and outlet boundary condition.

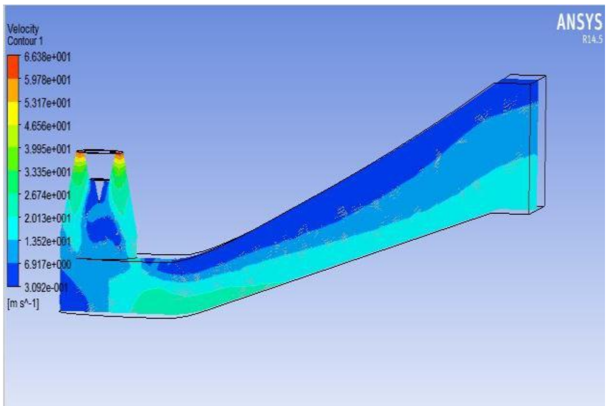


Fig. 2. Velocity contour.

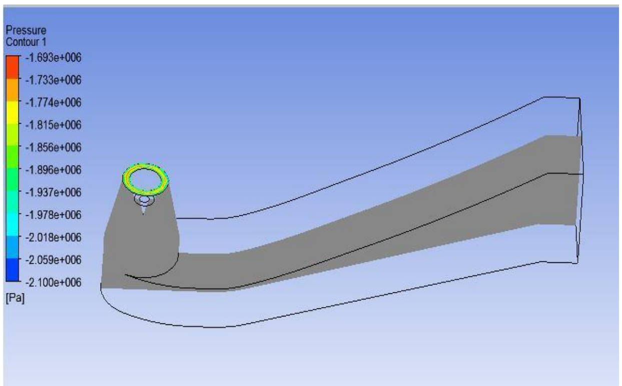


Fig. 3. Pressure at inlet.

Shows the pressure at inlet(-2.100×10^5Pa) and pressure at outlet(1.071×10^5 Pa)respectively. [1]

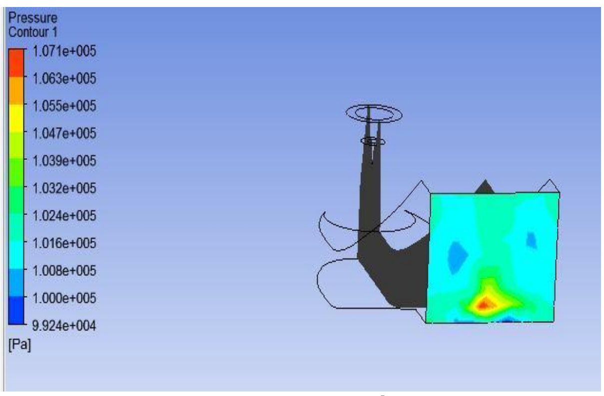


Fig. 4. Pressure at outlet.

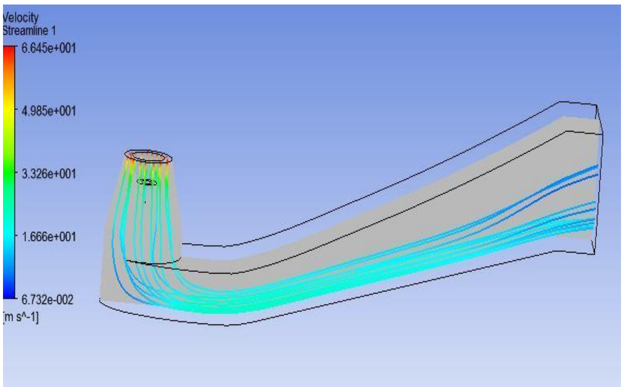


Fig. 5. Velocity streamline.

Comparison with Experimental Reading

The pressure and velocity at inlet and outlet of draft tube has been measured by experimental procedure. The same results have been compared with ANSYS analysis results and % difference has been found as given in table 1, which shows both results are in good agreement with each other.

Comparison between ANSYS and practical reading

	Inlet Pressure	**Outlet Pressure**
Ansys Result	-2.100×10^5Pa	1.071×10^5 Pa
Practical Reading	-1.99×10^5Pa	1.12×10^5 Pa
% Difference	5.23 %	4.38

III. LITERATURE REVIEW

(a). Design modification of draft tube for hydro power plant using CFD analysis.

Investigators	Mechanism Used	Result & Discussion
Gunjan B. Bhatt, Dhaval B.Shah, Kaushik M. Patel	Design the Draft Tube with the help of CFD analysis (Design in CREO & analysis in CFX solver)	1. CFD analysis of Draft tube and Find pressure and velocity distribution which are approximately matching with experimental readings. 2. Compare five different concepts of draft tube and compare its efficiency.

(b). Numerical simulation for performance of Elbow draft tube at different geometrical configurations.

Investigators	Mechanism Used	Result & Discussion
Vishnu Prasad, Ruchi khare, Abhas chincholikar	Design the Draft Tube with the help of CFD analysis (Design in CREO & analysis in CFX solver)	1. CFD analysis of Draft tube and Find pressure and velocity distribution which are approximately matching with experimental readings. 2. Compare five different concepts of draft tube and compare its efficiency.

- During 1909 to1929, number of investigations were carried out on straight diffuser. The work of A. Gibson, H. Hochschild and I. Nikurdze refers to this field. H.Hochschild and I. Nikurdze observed the coefficient of resistance and characteristics of flow. Later in USA the idea of bell mouthed draft tube in the nineteenth century comes in consideration but its practical application found latter. In 1917, W.M. White designed and built a bell mouthed tube for the hydraulic stations at Niagara Falls. In 1919, moody suggested that the well mouthed tube with a cone that would fill the dead zone created in the swirling flow. The moody draft tube was widely used in large hydroelectric stations in USA.
- A lot of work and studies have been done on turbines (existing and newly developed) using Computational Fluid Dynamics which described in various research papers. Few of the papers which are referred are described below.

IV. METHODOLOGY

Various Steps Used For CFD Analyses of Elbow Draft Tube :-

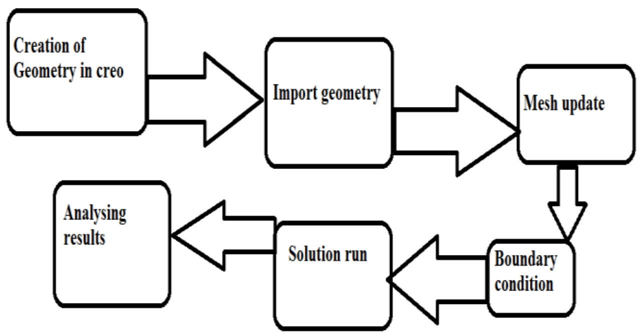


Fig. 6. Steps used in CFD simulation.

Software Work in Three Processes

{a} Pre-processor.
{b} Simulation.
{c} Post-processor.

- A pre-processor for creating, assembling, or modifying the data required to analyze the simulation, and for generating the required database file.
- A simulation engine for performing the numerical calculations required to analyze the process, and writing the results to the database file. The simulation engine reads the database file, performs the actual solution calculation, and appends the appropriate solution data to the database file.
- A post-processor for reading the database file from the simulation engine and displaying the results graphically and for extracting numerical data.

V. FUTURE OBJECTIVE

The draft tube plays an important role on overall performance of reaction turbine. The efficiency of draft tube depends on its shape and other geometric parameter the most commonly used draft tube is elbow draft tube with rectangular outlet. The shape and area of draft tube at outlet as well as along its length may improve the efficiency of turbine.

VI. Conclusion

In this research our main objective is to design a elbow draft tube with varying diffuser angle and to find out the optimized model for the same or with enhanced diffuser angle. The simulation method is used to resolve the problem of draft tube design and to reduce the cost of prototyping and to get the improved results by using computational fluid analysis in ANSYS 14.0 by CFX solver. And determining pressure and velocity profile at inlet and outlet condition for different diffuser angles like 0^0, 10^0, 20^0, 30^0 degrees. We are considering four cases of elbow draft tube to improve efficiency and pressure at outlet. The same analysis has been performed for different Cases with the same boundary conditions.

References

[1] G. Bhatt, D. B. Shah, and K. M. Patel, "Design automation and CFD analysis of draft tube for hydro plant," *International Journal of Mechanical and Production Engineering*, vol. 3, issue 6, 2015.

[2] U. Anderson, "Experimental study of sharp heel draft tube," Thesis (PhD), Lulea University of Technology, Sweden, 2009.

[3] R. Khare, V. Prasad, and S. Kumar Mittal, "Effect of runner solidity on performance of elbow draft tube," *Proceedings of the 2nd International conference on advances in energy engineering (ICAEE), Energy Procedia*, 14, pp. 2054-2059, 2012.

[4] V. Prasad, R. Khare, and A. Chincholikar, "Hydraulic performance of elbow draft tube for different geometric configurations using Cfd," *IGHEM -2010*,Oct. 21 -23, AHEC, IIT Roorkee India, 2010.

[5] U. Rajak, V. Prasad, and R. Khare, "Numerical flow simulation using star CCM+.," *IISTE*, vol. 3, no. 6, 2013.

[6] A. Siake, C. Koueni-Toko, B. Djemako, and T.-Toko, "Hydrodynamic characterization of draft tube flow of a hydraulic turbine," *IJHE-2014*, pp. 103-114, 2014.

[7] J. Obrovsk, "Development of high specific speed Francis turbine for low head," *Engineering Mechanics*, vol. 20, no. 2, pp. 139–148, 2013.

[8] C. B. Cook, M. C. Richmond, and J. A. Serkowski, "Observations of velocity conditions near a hydroelectric turbine draft tube exit using ADCP measurements," *Flow Measurement and Instrumentation*, Vol. 18, pp. 148–155, 2007.

[9] B. G. Mulu, P. P. Jonsson, and M. J. Cervantes, "Experimental investigation of a kaplan draft tube-part I best efficiency point," *Applied Energy*, vol. 93, pp. 695-706, 2012.

[10] M. Nishi and S. Liu, "An outlook on the draft-tube-surge study," *International Journal of Fluid Machinery and Systems*, vol. 6, no. 1, January-March 2013.