International Journal of Technology and Emerging Sciences (IJTES)

www.mapscipub.com

Volume 03 || Issue 04 || October 2023 || pp. 13-18 E-ISSN: 2583-1925

CFD Simulation and Analysis of Elbow Draft Tube for Varied Diffuser Angle Employing ANSYS

Anurag Kumar[1], Mukesh Kumar[2], Chandan Kumar[3]

[1]*Cambridge Institute of Technology, Department of Mechanical Engineering, Ranchi, Jharkhand, India*
[2] *Cambridge Institute of Technology, Department of Mechanical Engineering, Ranchi, Jharkhand, India*
[3] *Cambridge Institute of Technology, Department of Mechanical Engineering, Ranchi, Jharkhand, India*

---***---

Abstract: The display of a hydraulic reaction turbine's draught tube has a significant impact on its performance. The form and speed dispersion at the bay are crucial factors that affect how the draught tube is displayed. The organization of this section has typically been based on enhanced analytical techniques, test thumb rules, and model tests. Due to its versatility and economic viability, the application of computational liquid elements (CFD) in the planning process has grown dramatically over the last decade or so and will continue to do so.

The elbow draught tube has undergone CFD (ANSYS 15.0, CFX) analysis and optimization in the current work to estimate the pressure and velocity profile at inlet and outlet conditions. To increase effectiveness and outlet pressure, seven cases (ranging from 00 to 300) have been suggested for the draught tube. Maximum outlet pressure and velocity, which have been optimized, are 1.09×10^6 Pa and 72.27 m/s, respectively. In comparison to the base model, Case-5.2.1, diffuser angle=200, the optimized model is attained for the greatest outlet pressure and velocity with for diffuser angle=$20^0$with horizontal.

Key Words: Turbine, Diffuser angle, Computational Fluid Dynamics (CFD), Draft Tube

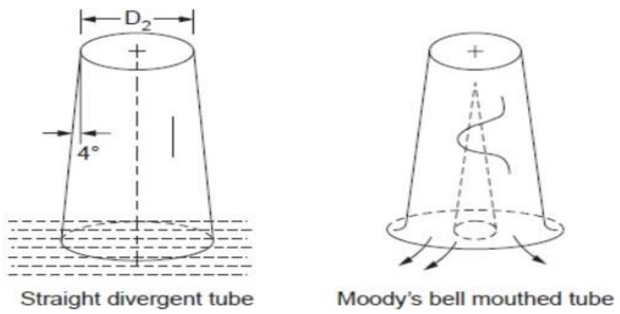
1. INTRODUCTION

1.1. Definition of Draft Tube

A water turbine's draught tube is designed to lower the exit velocity with the least amount of energy loss. The dynamic pressure (kinetic energy) is "converted" into static pressure via the draught tube. Not all of the energy will be recovered; losses cause the overall pressure to drop via the diffuser. The draught tube is a relatively simple geometrical object—it is a bending pipe that diverges in the direction of the stream. However, various unstable effects have been seen due to the complicated dynamical processes of the flow in a draught tube.

The draught tube is a crucial element in the design of a hydroelectric system that has a big impact on both efficiency and cost, especially in low-head systems. Due to how performance affects total efficiency, even a small improvement could lead to significant energy savings. More compact designs may be less expensive because draught tubes can be bulky and expensive. A detailed understanding of diffuser performance is necessary to achieve the best balance between efficiency and price. Although designers of conventional systems have a lot of expertise, there is always room for improvement.

Although there are a few minor design variations available, some design factors are more crucial than others. More often than not, the outlet area is more significant than the outlet's shape, whether circular or rectangular. However, one of the trickiest issues with draught tubes is how the elbow is shaped. The difficulty lies in changing the shape with minimal energy loss and without running the danger of harmful mechanisms such severe cavitation's. Previously, a few hydro-mechanical concepts were used to develop the draught tube, with careful consideration given to its structural and constructional application.



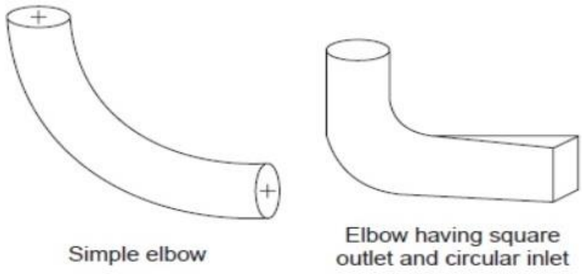


Fig. 1. Various shapes of draft tubes

1.2. Types of Draft Tube

For adjustable blade turbines, different types of draft tubes are in use depending on the power output and the orientation of the axis of rotation. The kinetic energy leaving the runner determines the dimensions of draft tubes for ensuring sufficient recovery of this energy. Thus, the cross-sectional area of the exit for all types of draft tubes must be 3 to 4 times the area of the cross-section at the inlet. Frequently, the choice of the type of the draft tube to be used is constrained by certain given conditions. Vertical adjustable blade turbines are in use in hydroelectric stations along with one of the following three types of draft tubes:

- Straight conical
- Curved draft Tube.
- Bell Mouth with or without cone.

2. COMPUTATION OF FLOW PARAMETERS

The following parameters are to be computed using the observed data from experimental test of turbine model:

(1) Net head can be evaluated by Eq.

$$H_n = \frac{TP_{CI} - TP_{DO}}{\gamma} \qquad 1$$

(2) Head utilized by the runner can be evaluated by Eq.

$$H_R = \frac{2NT}{60 * \gamma Q} \qquad 2$$

(3) Average flow velocity at inlet can be evaluated by Eq.

$$C_1 = \frac{Q}{A_1} \qquad 3$$

(4) Average flow velocity at the outlet can be evaluated by Eq.

$$C_2 = \frac{Q}{A_2} \qquad 4$$

(5) Net head on the turbine at the outlet can be evaluated by Eq.

$$H_n = \left(\frac{P_1}{\gamma} + \frac{C_1^2}{2g} + z_1 \right) - \left(\frac{P_2}{\gamma} + \frac{C_2^2}{2g} + z_2 \right) \qquad 5$$

(6) Unit speed can be evaluated by Eq.

$$n_{11} = \frac{nD}{\sqrt{H_n}} \qquad 6$$

(7) Unit discharge can be evaluated by Eq.

$$Q_{11} = \frac{Q}{D^2 \sqrt{H_n}} \qquad 7$$

(8) Input power can be evaluated by Eq.

$$P_{in} = \gamma Q H_n \qquad 8$$

(9) Output power can be evaluated by Eq.

$$P_{out} = \frac{2\pi nT}{60} \qquad 9$$

(10) Hydraulic efficiency of turbine can be evaluated by Eq.

$$\eta_H = \frac{P_{out}}{P_{in}} * 100 \qquad 10$$

(11) Speed factor can be evaluated by Eq.

$$SF = \frac{nD}{\sqrt{gH_n}} \qquad 11$$

(12) Discharge factor is evaluated by Eq.

$$DF = \frac{Q}{D^2 \sqrt{gH_n}} \qquad \mathbf{12}$$

3. LITRATURE REVIEW

Anderson [2] In their work, have done a surrogate-based optimization (SBO) framework, in order to develop and implement a computer tractable approach was used to optimize the shape of hydraulic turbine draft tubes.

By using this methodology, one can explore the design and solution space more quickly and effectively by substituting a less expensive surrogate model for the costly CFD model during the optimization phase. using a numerical approach for calculation and a CFD approach for modelling and analysis.

Draught tubes for hydraulic turbines have been constructed, and by Khare et al. [3] have investigated the parallel performance of commercial CFD software on homogeneous computer networks.

Results from the CFX-5.7.1 stable and unsteady CFD simulations. Additionally, there was no discernible difference between the turbulence model and the applied inlet boundary conditions.

A verified numerical simulation approach to assess the performance of global draught tubes was provided by Prasad et al. in 2010 [4].

Designers can dependably utilize this method, which is based on steady-state flow simulations utilizing the k-turbulence model and a moderately improved mesh, to compare the relative global draught tube performance of close-by design operating points.

This study emphasizes the significance of selecting turbulent inlet boundary conditions, even those that are near the operating condition with the highest efficiency.

Using the ANSYS CFX code, Vishnu et al. [4] carried out a numerical flow simulation for a 3D viscous turbulent flow in an elbow draught tube by altering its length and height at various mass flow rates. To examine how geometrical factors, affect draught tube performance, the efficiency and losses of the tubes are computed using pressure and velocity distributions and graphically displayed.

The geometry of the draught tube employed in the majority of hydroelectric power plants is compatible with the geometrical parameters predicted by numerical simulation for the best performance.

For an experimentally tested turbine, Rajak et al. [5] performed a 3-Dimensional (3-D) real flow analysis, and the features of the prototype turbine were predicted under real operating regimes. The running scenario was treated as a real prototype turbine, and the flow structure inside the device was examined. An improvement in the design of the casing tip piece was demonstrated by visualizing the findings in CFX-post, and the conclusions were confirmed by experimental data.

Using Open FOAM -1.5-dev, Siake et al. [6] presented a numerical simulation of the flow in the draught tube of the Kaplan turbine. A Kaplan Turbine's draught tube test case for the Turbine-99 was simulated. The outcomes are in line with the state of the art as described in the literature. The flow simulation effectively captures the flow's overall depiction. However, more precise inlet conditions and turbulence models are required to employ the CFD for quantitative assessments of efficiency or local behavior.

The conical draught tube for hydraulic turbine optimization model is shown by Khare et al. [7].

In this research, conical draught tubes are numerically simulated for various lengths and diffuser angles using the CFD code ANSYS CFX , by computing head loss, head recovery, and draught tube efficiency for simulation results, the performance of the draught tube is examined.

Obrovsk [8] done other model of draft tube for numerical simulation. In this paper, unsteady flow calculation made by various intervals and presented in tabular and graphical form STAR CCM+ software is used during model making process and gives inherent results.

Christopher et al. [9] again numerical analysis done on different model of draft tube. Special attention on friction effect through the flow inside the complex geometry of draft tube.
Comparison of previous Anderson [2] results and give various graphs on different rpm and validate the results to get efficient results.

According to Mulu et al. [10], the elbow draught tube with dividing pier numerical simulation is most effective when the draught tube is L= 10 * D1 in length.

Comparative analysis of the pressure variations and velocity contours at the draught tube's inlet section and immediately following the elbow section reveals that the location of the dividing pier has a substantial impact on the velocity distribution.

4. ANALYSIS

Step-1: For the creation of geometry, we select the design as given in Figure 4.2, and the author has created it with the help of CAD software, i.e., CREO, and then the geometry is to be imported into ANSYS Software by using the CFX tool [11-16].

The elbow draught tube model is meshed in step 2. Figure 2 illustrates the use of ten node tetrahedral elements for fine meshing of the CFD mesh type.

This element was chosen because it provides excellent meshing on elbow draught tube curvature portions, as demonstrated in fig 2.

In the meshing model, the values of 9030 nodes and 46401 elements are to be generated.

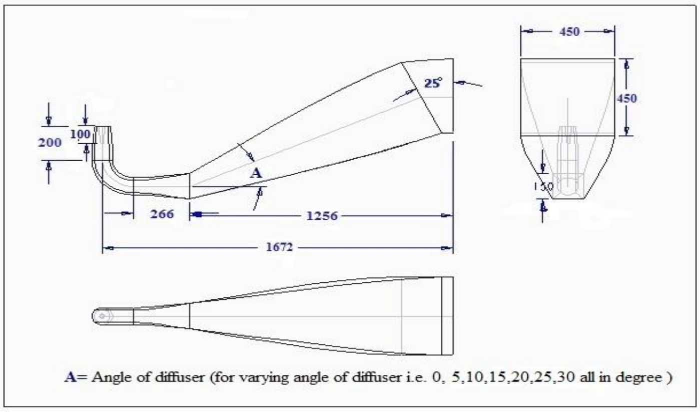


Fig. 2. 2D drawing of elbow draft tube

The ANSYS 15.0 CFX solver at the postprocessor step determines the pressure and velocity distribution. The results for the Elbow draught tube's velocity and pressure contour are depicted in Figures 3 and 4, respectively.

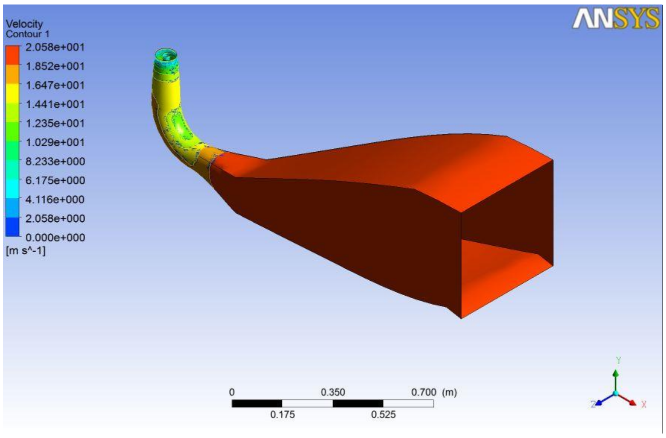


Fig. 3. Velocity Contour of Elbow Draft Tube for Base model at (diffuser angle 20^0)

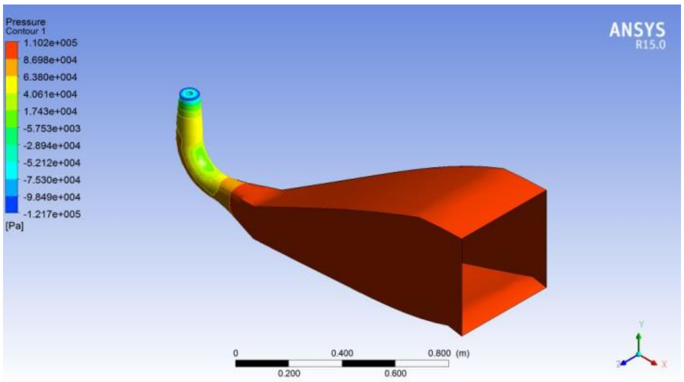


Fig 4. Pressure Contour of Elbow Draft Tube for Base model (diffuser angle 20^0)

For the maximum value of outlet pressure, a graph has been created in order to compare all present cases, which have been taken for the analysis as shown in Figure 5.

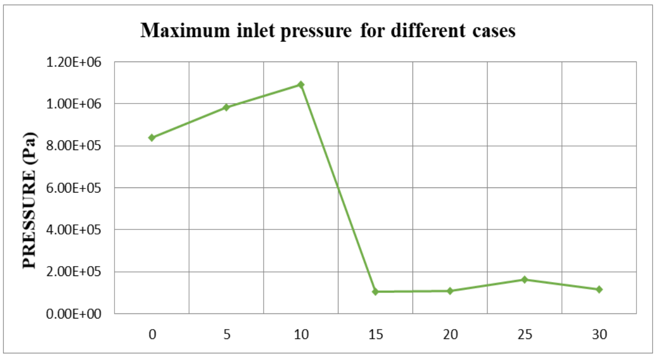


Figure 5. Line chart for outlet Pressure for different values of diffuser angle

The highest output pressure achieved in Case-5.2.4, or diffuser angle 10 0 from the horizontal and its value are given in Figure 6 of the output pressure line chart. Table 3 provides the maximum and minimum outflow and inlet velocities for each Case.

For the maximum value of outlet Velocity, a graph has been created in order to compare all cases that have been considered for the present analysis as shown in Figure 6.

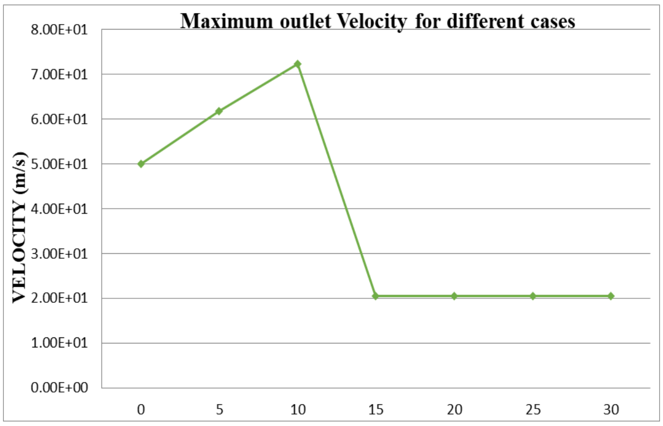


Figure 6. Line chart for outlet velocity for different diffuser angle

5. CONCLUSION

The elbow draught tube has undergone CFD (ANSYS 15.0, CFX) analysis and optimization in the current study to estimate the pressure and velocity profile at inlet and outlet conditions.

(1) To increase effectiveness and outlet pressure, seven cases (ranging from 00 to 300) have been proposed for draught tubes.

(2) Different Cases with the same boundary conditions have undergone the same analysis. According to analysis results compared to all other examples and the base model, Case 5.2.1,

Case 5.2.4 achieve the enhanced value of maximum pressure and velocity.

(3) The realized input pressure and output pressure are 2.51% and 1.78%, respectively, indicating good agreement and an acceptable range between the current work (ANSYS 15.0 CFX) and experimental reading.

(4) 1.09×10^{6} Pa and 72.27 m/s are the optimal maximum outflow pressure and velocity values.

(5) Compared to the base model, Case-5.2.1, diffuser angle=20^{0}, the optimized model is attained for the greatest outlet pressure and velocity with Case-5.2.4, for diffuser angle=10 0 with horizontal.

(6) The experimental method and ANSYS work by reference to [Gunjan B. Bhatt et. al.[1] have been used to analyze the pressure distribution at the intake and outflow of the draught tube. When the same results for an elbow draught tube are compared to the current work in ANSYS (CFX), they show good agreement and an acceptable range with one another. As a result, the current analysis may be used to avoid more expensive experimentation.

Therefore, it can be said that CFD analysis is a very efficient method for accurately simulating numerical flow in complex flow fields.

REFERENCES

[1] G. Bhatt ,D. B. Shah, K. M. Patel, Design Automation And CFD Analysis of Draft Tube For Hydro Plant, International Journal of Mechanical And Production Engineering ,2015, Vol. 3 (6).

[2] Anderson U., Experimental study of Sharp Heel Draft Tube, Thesis (PhD), Lulea University of Technology, 2009. "ISBN: 978-91-86233-68-6, Sweden.

[3] Ruchi Khare, Vishnu Prasad, Sushil 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, 2012, page 2054-2059.

[4] Vishnu Prasad, Ruchi Khare, Abhas Chincholikar, Hydraulic Performance of Elbow Draft Tube for Different Geometric Configurations Using CFD, IGHEM, 2010, Oct.21 -23,AHEC,IIT Roorkee India.

[5] Upendra Rajak , Vishnu Prasad ,Ruchi Khare, Numerical flow simulation Using Star CCM+." IISTE, Vol.3 (6), 2013.

[6] Siake A, Koueni-Toko C., Djemako B.,Tcheukam-Toko, Hydrodynamic Characterization of Draft Tube Flow of a Hydraulic Turbine.IJHE, 2014, pp.103-114.

[7] Ruchi Khare, Vishnu Prasad, Mitrasen Verma, Design Optimisation of Conical Draft Tube of Hydraulic Turbine. International Journal of Advances in Engineering, Science and Technology, Vol. 2 (1), 2012.

[8] Jiri Obrovsk, Development of high specific speed Francis turbine for low head Engineering Mechanics, Vol. 20 (2), 2013, pp. 139–148.

[9] B. C. Christopher, M. C. Richmond, John A. Serkowski, Observations of Velocity Conditions Near a Hydroelectric Turbine Draft Tube Exit Using ADCP Measurements, Flow Measurement and Instrumentation, 2007, Vol. 18 , pp.148–155.

[10] B.G. Mulu, P.P. Jonsson, M.J. Cervantes, Experimental Investigation of a Kaplan Draft Tube-Part I Best Efficiency Point", Applied Energy, 2012, Vol. 93, pp.695 - 706.

[11] Michihiro Nishi and Shuhong Liu, An Outlook on the Draft-Tube-Surge Study, International Journal of Fluid Machinery and Systems, 2013, Vol. 6, No. 1.

[12] Thi C. Vu, Christophe Devals, Ying Zhang, Bernd Nennemann and François Guibault, Steady and unsteady flow computation in an elbow draft tube with experimental validation. International Journal of Fluid Machinery and Systems, 2011, Vol. 4 (1).

[13] I.Gunnar , J. Hollestrom, Redesign of Existing Hydro Power Draft Tube. ISSN 1402-1617,Sweden.

[14] Coelho J. G., Brasil Junior A. C. P. Numerical simulation of draft tube flow of a bulb turbine, International Journal of Energy and Environment, Vol.4, Issue 4, 2013 pp.539-548.

[15] Dheeraj Sagar, Tarang Agarwal, Shubham Bhatnagar, Recapitulation of Draft Tubes. International Journal of Advance Research In Science And Engineering IJARSE, 2015, Vol.4 (01).

[16] Marjavaara B. D, CFD driven Optimization of Hydraulic Turbine Draft Tubes Using Surrogate models, 2006, Thesis (PhD), Lulea University of Technology ISSN: 1402-1544, Sweden.

Table 1. Experimental Reading, ANSYS (CFX), and the Present are Compared with Gunjan B. Bhatt et al.[1]

Comparison	Inlet Pressure (Pa)	Outlet Pressure(Pa)
Present work in ANSYS (CFX) With Elbow	1.21×10^{5}	1.10×10^{5}
Present work in ANSYS (CFX) (Without Elbow)	2.100×10^{5}	1.071×10^{5}
Experimental Work [Bhatt et.al.[1]	1.99×10^{5}	1.12×10^{5}
% Deviation in between Present and Experimental work	2.51	1.78

Table 2: Pressure at inlet for different angle Cases

Case No	1	2	3	4	5	6	7
Pressure	Case 5.2.2 (0^0)	Case 5.2.3 (5^0)	Case 5.2.4 (10^0)	Case 5.2.5 (15^0)	**Case 5.2.1 (20^0) BASE MODEL**	Case 5.2.6 (25^0)	Case 5.2.7 (30^0)
Maximum (inlet) in Pa	8.38×10^{5}	9.82×10^{5}	1.09×10^{6}	1.05×10^{5}	1.102×10^{5}	1.61×10^{5}	8.11×10^{5}

Case No	1	2	3	4	5	6	7
Pressure	Case 5.2.2 (0^0)	Case 5.2.3 (5^0)	Case 5.2.4 (10^0)	Case 5.2.5 (15^0)	**Case 5.2.1 (20^0) BASE MODEL**	Case 5.2.6 (25^0)	Case 5.2.7 (30^0)
Minimum (inlet) in Pa	1.44×10^{6}	1.75×10^{6}	2.45×10^{6}	1.21×10^{5}	1.217×10^{5}	7.2×10^{4}	1.217×10^{5}

Table 3: Velocity at the Outlet and Inlet for Different angles of diffuser

Case No	1	2	3	4	5	6	7
Velocity	Case 5.2.2 (0^0)	Case 5.2.3 (5^0)	Case 5.2.4 (10^0)	Case 5.2.5 (15^0)	**Case 5.2.1 (20^0) BASE MODEL**	Case 5.2.6 (25^0)	Case 5.2.7 (30^0)
Maximum (Outlet) in (m/s)	49.99	61.72	72.27	20.53	20.58	20.49	20.49