

CFD Simulation and Analysis of Conical Draft Tube of Experimental Setup of Hydraulic Turbine

Sumeet J. Wadibhasme¹, Shubham Peshne², Pravin Barapatre³, Santosh Barade⁴, Prof. Shailendra Daf⁵,
Prof. Amit Puttewar⁶

U. G. Student, Department of Mechanical Engineering, Priyadarshini Bhagwati College of Engineering, Rashtasant
Tukdoji Maharaj Nagpur University, Maharashtra, India^{1,2,3,4}

Asst. Professor, Department of Mechanical Engineering, Priyadarshini Bhagwati College of Engineering, Rashtasant
Tukdoji Maharaj Nagpur University, Maharashtra, India^{5,6}

ABSTRACT: The design of a draft tube significantly affects the efficiency of mixed and reaction flow turbine setup. The purpose of providing the draft tube is to recover the part of kinetic energy coming out of runner. Many researches have been made to compare and optimize the design of draft tube to minimize the head loss. In this study, we have put our efforts to study the working of draft tube and further carry out CFD simulation to study the flow pattern inside the conduit and results are analyzed for the existing design of draft tube of the experimental setup of hydraulic turbine. The pressure difference between inlet and outlet is computed.

KEYWORDS: -Draft tube, elbow, hydropower, pressure head, efficiency, swirl, tailrace, turbine.

I. INTRODUCTION

The draft tube is one of the important component of a hydro machinery setup. The efficiency of each component of turbine affects the performance of hydro-power plant. The efficiency of draft tube also has a considerable effect on the efficiency of turbine setup. The draft tube is a conduit which connects the runner exit to the tailrace where the water is being finally discharged from the turbine. The main purpose of draft tube is to allow the installation of turbine above the tailrace level. This permits the turbine to have access to the water without being immersed in water. The hydrodynamic investigation on straight conical draft tube were carried out since 1903 and investigators adopted walls of draft tube parallel to streamline based on theoretical solutions. However, this was found to be impractical as the draft tube fluid flow is difficult to predict numerically due to the fluctuating, non-uniformity and swirled flow that leaves the rotating runner. After passing through the turbine runner, the exiting fluid still has appreciable kinetic energy, and perhaps swirl. To recover some of this kinetic energy which would otherwise be wasted, the flow enters an expanding area of the diffuser called as draft tube, which turns the flow horizontally and slows down the flow speed, while increasing the pressure prior to discharge into the tailrace.

The objective of present work is to study the draft tube and the important phenomenon taking place in the flow path by considering the study carried out in available literatures on the various types of draft tube. In this paper, CFD is used as a tool to simulate the fluid flow through the draft tube. The loss, head recovery and efficiency characteristics of the available design of draft tube used in the Hydraulic experimental setup of Kaplan turbine in our institution have been predicted at off design condition from simulation results. The inlet conditions are assumed by comparison with the literature data available on basis of net head and other parameters due to lack of experimental data.

II. PRINCIPLE OF DRAFT TUBE

The principle of draft tube can be represented mathematically by help of Bernoulli's equation between section 2-2 and 3-3.

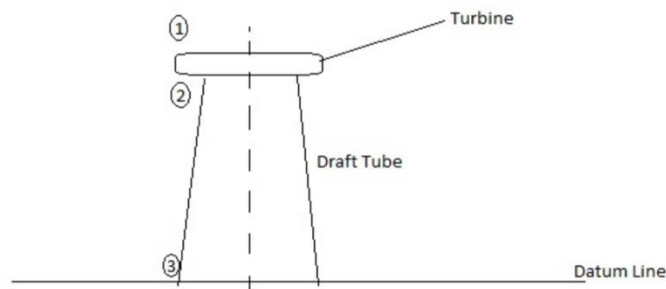


Fig1: schematic representation of section of draft tube

$$z_2 + \frac{P_2}{\rho g} + \frac{V_2^2}{2g} = z_3 + \frac{P_3}{\rho g} + \frac{V_3^2}{2g}$$

where, P= Absolute pressure,
z= Height,
V= Mean velocity.

Difference in pressure head,

$$\frac{P_3 - P_2}{\rho g} = \frac{V_2^2 - V_3^2}{2g} + (z_2 - z_3).$$

Thus it is clear that the pressure at section 2-2 is below atmospheric pressure.

III. LITERATURE REVIEW

For design of any component it is necessary to know the flow physics occurring inside the component. To design an optimum draft tube is a challenging task as many flow phenomenon takes place simultaneously. Dr. Ruchi Khare, Dr. Vishnu Prasad, Mitrasen Verma had worked on conical draft tube of hydraulic turbine. The study showed the flow and head recovery of draft tube. In the study the performance of draft tube was analyzed by calculating head loss, head recovery coefficients and efficiency of draft tube from simulation results. The head recovery is more influenced at smaller length but has very negligible change after the length equal to $19D_3$. [2]

Gunjan B. Bhatt, Dhaval B. Shah, Kaushik M. Patel worked on elbow draft tube of hydro power plant. Their study attempted for design automation of modelling of draft tube using Excel spreadsheet and Creo parametric software. Microsoft Excel analysis tool transfers the spreadsheet data to the Creo database. The ANSYS workbench was used for CFD analysis of draft tube. The concept design with sharp heel draft tube was extended with radius at both sides which gave maximum outlet pressure as per analysis results compared to all other concepts. 5.23 % difference in inlet pressure between ANSYS result and practical reading; 4.38 % difference was observed in outlet pressure. [3]

Dr. Vishnu Prasad, Dr. Ruchi Khare, Abhas Chincholikar have worked on performance of elbow draft tube at different geometric configurations. They referred Volozsky hydro-electric station where the draft tube of height $2.24D_1$ is being used instead of $1.915D_1$ to obtain an additional power output of 100-150 KWhr for initial analysis. Shear Stress Transport (SST) turbulence model in ANSYS CFX code was used for analysis due to boundary curvature

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 4, April 2016

in elbow and diffusing flow. It was observed that the mass flow rate has no effect on efficiency and loss characteristics of draft tube for constant L/D_1 ratio because of increase in kinetic energy at inlet with increase in mass flow rate, head recovery and losses. [4]

A study on draft tube for high head Francis turbine. Was carried out by Vishal Soni, Amit Roghelia, Jaymin Desai, Vishal Chauhan. The turbine design initiated by CFD at Jyoti Ltd., project 'Swayam' was optimised. This initial design of draft tube was found to have large swirl at downstream of runner which resulted in more hydraulic losses. The design was modified by simulating various designs of bend type curved draft tube using CFD and optimum results were obtained in which the swirling was quite low and having smooth streamline flow. [5]

Z. Carija, Z. Mrsa and L. Dragovic has studied the hydraulic turbine flow simulations to measure the engineering quantities like power, torque, flow rate, efficiency etc. using CFD in the Holleforsen hydropower draft tube model. The turbine model was mounted in VAUB'S turbine rig at the Alvkarleby Laboratory in Sweden. Larger deviations from experimental results was observed in elbow part of draft tube because this is a region with highly complex fluid flow structure where RANS turbulence models are probably not enough appropriate for fluid flow prediction. After the elbow in horizontal draft tube part calculated pressure recovery almost equals to measurements. [6]

Shake A, Koueni-Toko C, Djeumako B, Tcheukam-Toko D, Soh-Fotsing B, Kitchen A. worked on effect of friction through the flow inside the complex geometry of the draft tube and for the interaction between the vortex structure and the draft tube volute. This study was done with reference of Andersson U. doctoral thesis based on experimental study of sharp-heel Kaplan draft tube. The study presented the pressure and velocity fields showing the separate zones and the zones of recirculation which influence the flow in the draft tube of hydraulic turbine. [7]

IV. GEOMETRICAL MODELLING OF DRAFT TUBE

Simple Conical draft tube is a diffuser pipe. The geometry of draft tube was created in ANSYS WORKBENCH 14.0. The geometric parameters of the respective selections are shown in Fig: 2.

Table 1: Geometrical Parameter

Sr. No.	Feature Name	Dimension
1	Inlet diameter	150 mm
2	Outlet diameter	300 mm
3	Length	500 mm
4	Diffuser angle	8.53°
5	Hub diameter	78 mm

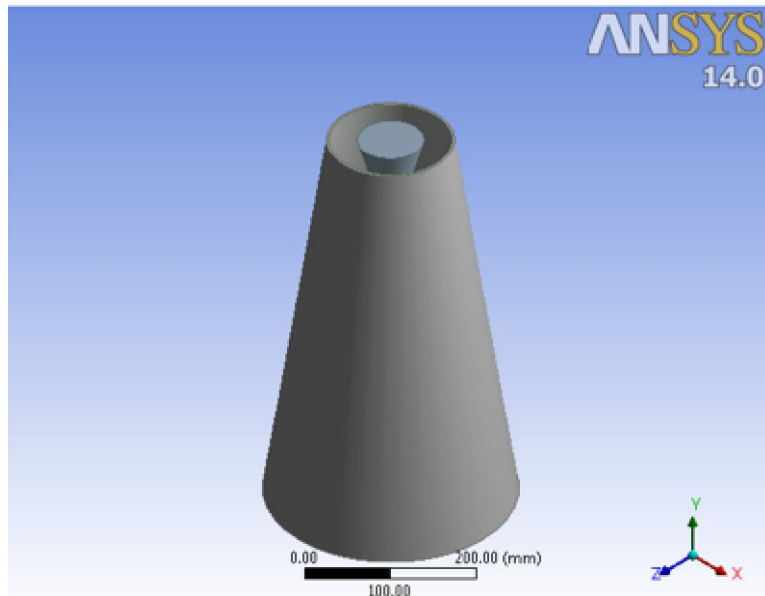


Fig 2: Geometric Model of Draft Tube

V. CFD ANALYSIS

Here ANSYS WORKBENCH 14.0 is used for CFD simulation and analysis of draft tube. The geometry was fine meshed using Tetrahedral and Hexahedral elements. The reason for selecting these elements is to obtain good meshing. Meshed model consists of 50,763 nodes and 1,66,502 elements. The fine mesh is generated near the surfaces in order to capture boundary layer as compared to interior.

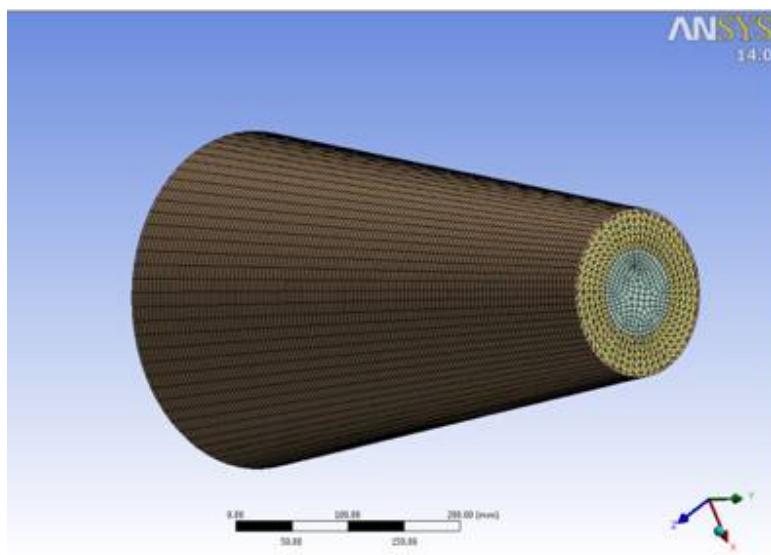


Fig 3: Meshing of Draft Tube.

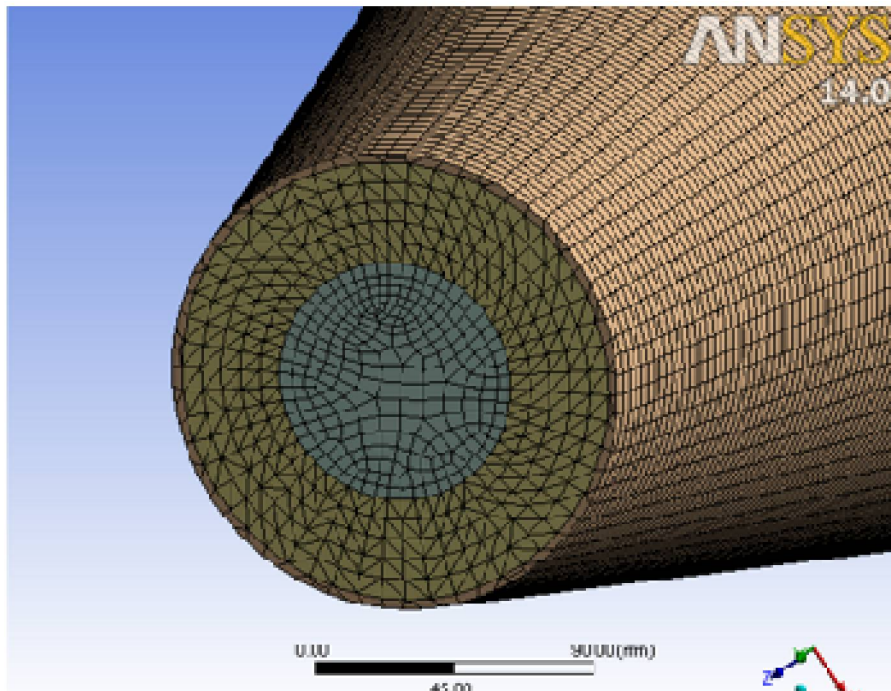


Fig 4: Close view of meshed part (Inlet) of Draft Tube.

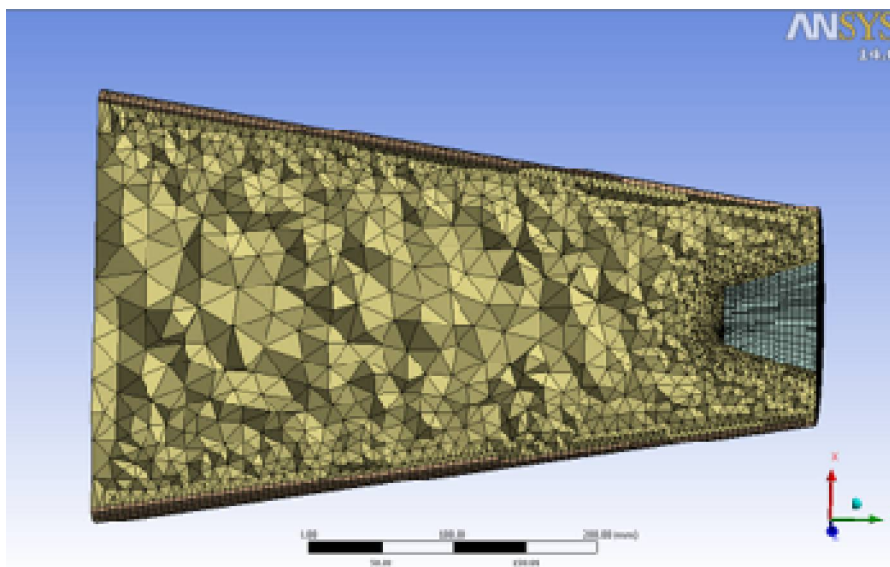


Fig 5: Section plane of mesh.

The boundary conditions of draft tube analysis have been applied. In boundary condition cylindrical inlet velocity components as, Axial component = 3.5 m/s, Tangential component = 2.43 m/s were applied and at outlet 0 gauge pressure (1 atmospheric pressure) is given [8].

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 4, April 2016

VI. CFD ANALYSIS

The design was simulated by CFD and on investigating the design a large amount of swirl and flow separation leading to bubble formation called as cavitation. was found at downstream of turbine runner. This presence of such large swirling is due to inefficient design of draft tube. It is clear that due to large swirling of flow at inlet of draft tube more hydraulic losses. The pressure at inlet (-3.388×10^5 Pa) and at outlet (0.0249×10^5 Pa) are found.

After that the results are analyzed and visualized in post processor as shown in Fig. 6 to Fig. 8.

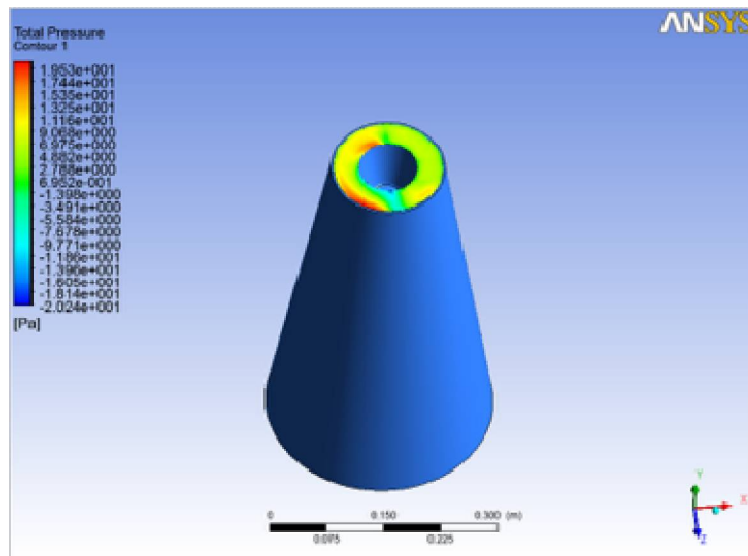


Fig 6: Pressure at Inlet.

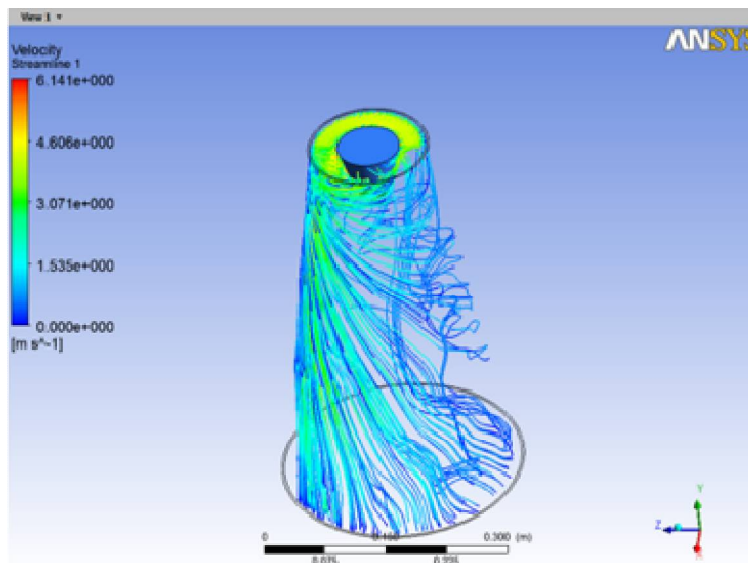


Fig 8: Velocity Streamline.

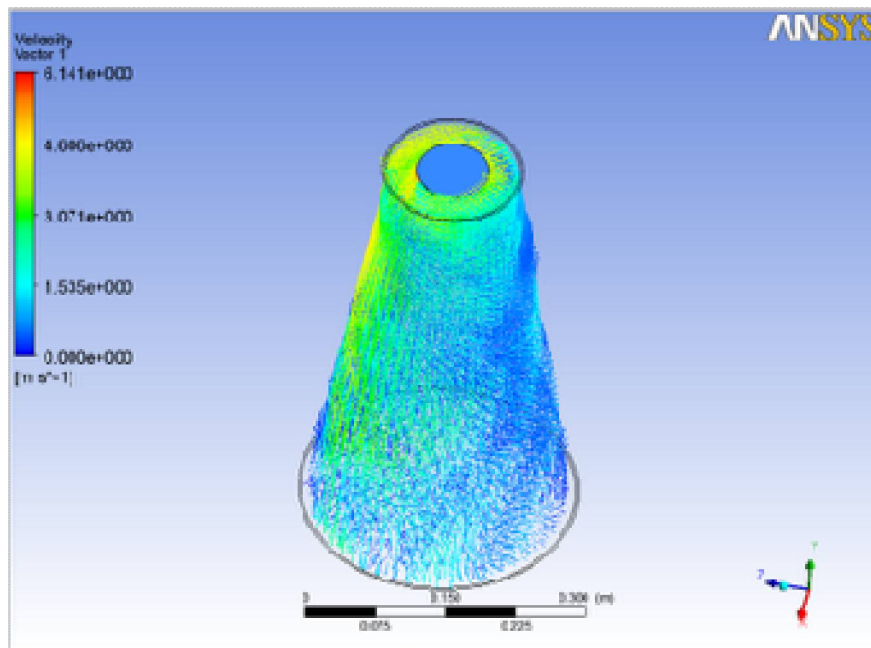


Fig 9: Velocity Vectors of flow.

VII. CONCLUSION

It is seen from this study ANSYS is very important tool for geometric modelling and analysis purpose and CFD is an effective tool for fluid flow simulation in complex flow domain with reasonable accuracy. The draft tube in our design consideration possesses large losses and flow separation leading to cavitation. Hence this design can be optimized further considering the obtained parameters as the reference values.

REFERENCES

- [1] Sumeet Wadibhasme, Shubham Peshne, Pravin Barapatre, Santosh Barade, Saurabh Dangore, Shubham Harde, Prof. Shailendra Daf, "HYDRAULIC TURBINE DRAFT TUBE: LITERATURE REVIEW", International Journal of Science, Engineering and Technology Research (IJSETR), Volume 5, Issue 3, March 2016, pp 673-676.
- [2] Fluid Mechanics Fundamental and Applications, Second Edition, Yunus A. Cengel & John M. Cimbala, Tata Mc GrawHill Publication, Page 816.
- [3] Dr. Ruchi Khare, Dr. Vishnu Prasad, Mitrasen Verma, "Design Optimization Of Conical Draft Tube Of Hydraulic Turbine", International Journal Of Advances In Engineering, Science And Technology (IAEST), ISSN: 2249-913x, Vol. 2 No. 1 Mar-May 2012.
- [4] Gunjan B. Bhatt, Dhaval B. Shah, Kaushik M. Patel, "Design Automation and CFD Analysis Of Draft Tube For Hydro Power Plant", International Journal Of Mechanical And Production Engineering, ISSN: 2320-2092, Volume- 3, Issue-6, June-2015.
- [5] Vishnu Prasad, Ruchi Khare, Abhas Chincolikar, "Numerical Simulation for Performance of Elbow Draft Tube at Different Geometric Configurations"
- [6] Vishal Soni, Amit Roghelia, Jaymin Desai, Vishal Chauhan, "Design Development of Optimum Draft Tube For High Head Francis Turbine Using CFD", 37th International & 4th National Conference On Fluid Mechanics And Fluid Power, FMFP2010, December 16-18, 2010, IIT Madras, Chennai, India.
- [7] Z. Čarija, Z. Mrša, L. Dragović, "Turbulent Flow Simulation In Kaplan Draft Tube", 5th International Congress Of Croatian Society Of Mechanics (ICCSM), September, 21-23, 2006, Trogir/Split, Croatia.
- [8] Shake A, Koueni-Toko C, Djeumako B, Tcheukam-Toko D, Soh-Fotsing B, Kitchen A., "Hydrodynamic Characterization of Draft Tube Flow of a Hydraulic Turbine", International Journal of Hydraulic Engineering 2014.
- [9] Dr. Vishnu Prasad, Dr. Ruchi Khare, "CFD: An Effective Tool for Flow Simulation in Hydraulic Reaction Turbines", International Journal of Engineering Research and Application (IJERA), ISSN: 2248-9622, Vol. 2, Issue 4, July-August 2012, pp. 1029-1035.
- [10] V De Henau, F A Payette, M Sabourin, C Deschenes, J M Gagnon, P Gouin, "Computational Study of A Low Head Draft Tube And Validation With Experimental Data", 25th IAHR Symposium on Hydraulic Machinery and Systems, IOP Conf. Series: Earth and Environmental Science 12(2010) 012084, DOI: 10.1088/1755-1315/12/1/012084.
- [11] CFD Driven Optimization of Hydraulic Turbine Draft Tubes using Surrogate Models, Doctoral Thesis, B. Daniel Marjavaara, Luleå University of Technology Department of Applied Physics and Mechanical Engineering Division of Fluid Mechanics, 2006:4I, ISSN: 1402-1544.