# HYDRAULIC TURBINE DRAFT TUBE: LITERATURE REVIEW

**Sumeet J. Wadibhasme[1], Shubham Peshne[2], Pravin Barapatre[3], Santosh Barade[4], Saurabh Dangore[5], Shubham Harde[6], Prof. Shailendra Daf[7]**

***Abstract*— The draft tube is one of the important components in a hydraulic mixed and reaction turbines. Without this the pressure at the outlet could drop as a result of lack of water which may adversely affect the efficiency of plant and may even fail to produced desired power. The study of flow patterns at inlet is very important for the effective knowledge about the performance of draft tube. In this study, we have discussed the principle of draft tube and its types through literature review. The parameter affecting the performance of draft tube are also discussed with the help of researches carried out earlier. The type of methods involved in the analysis of performance of draft tube are also considered from the literature available.**

***Keywords*—Draft tube, elbow, hydropower, pressure head, efficiency, swirl, tailrace, turbine.**

## I. INTRODUCTION

The draft tube is a conduit which connects the runner exit to the tailrace where the water is being finally discharged from the turbine. 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 draft tube is very important component in mixed flow and hydraulic turbines. The efficiency of each component of turbine affects the performance of hydro-power plant. The draft tube allows the turbine to be connected with tail race without being immersed in water. 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.

## II. DRAFT TUBE

The draft tube causes the pressure at the runner outlet to be lower than it would have been without the draft tube, by increasing the change in pressure from the inlet to the outlet of the turbine, it causes the pressure at the outlet of the runner to decrease below the atmospheric pressure, thereby enabling the turbine to utilize the available head most efficiently.

The turbine efficiency is based on Net Head (H) rather than Gross Head ($H_{gross}$). [1]

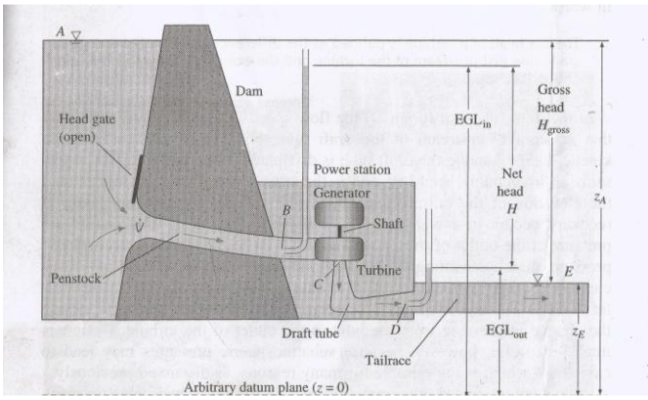


*fig: typical setup and terminology for a hydroelectric plant.*

$$\eta_{turbine} = \frac{W_{shaft}}{W_{water\ horsepower}} = \frac{bhp}{\rho g H V}$$

As Net Head H is head available at the output of draft tube, hence it is clear that the draft tube efficiency considerably affects the turbine efficiency.

The hydraulic characteristics of any draft tube depend on its shapes and dimensions and the flow pattern at its entrance. The increase in length of draft tube increases the frictional losses and thus reduce the draft tube efficiency. Similarly, the increase in diffuser angle will result in flow separation at the walls of draft tube and eddies may generate in the flow passage, which further increase the losses and reduce the efficiency. [2]

### A. TYPES OF DRAFT TUBE

1) Simple Conical draft tube- The draft tube has the shape of a frustum of a cone. This is generally provided for low specific speed. The cone angle is not to exceed 8°. For greater value of the cone angle it is seen that the flowing body of water may not touch the sides of the draft tube (Leaving the boundary). This will lead to the eddy formation bringing down the efficiency of the draft tube.

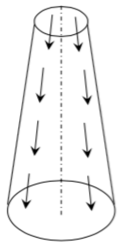

*fig: Simple Conical Draft Tube*



ISSN: 2278 – 7798 

2) Simple Elbow Draft Tube- The draft tube consists of extended elbow type tube used when turbine has to be placed close to tail race. It helps to cut down the cost of excavation and their exit diameter should be as large as possible to recover kinetic energy at the outlet of the runner. Such draft tubes are approximately 60% efficient.

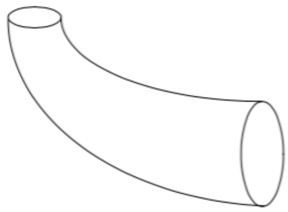

*fig: Simple Elbow Draft Tube*

3) Elbow Draft Tube with Varying Cross Section- This is further improvement of Simple Elbow Draft Tube. The outlet of draft tube should be situated below the tail race.

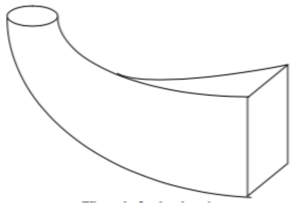

*fig: Elbow Draft Tube with Varying Cross Section*

4) Moody Draft Tube or Hydraucone- It is bell mouthed draft tube with a solid conical central core. The whirl of discharged water is very much reduced in this arrangement.

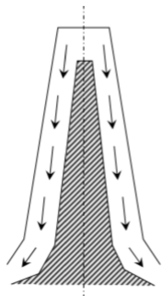

*fig: Moody draft tube*

### B. 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.

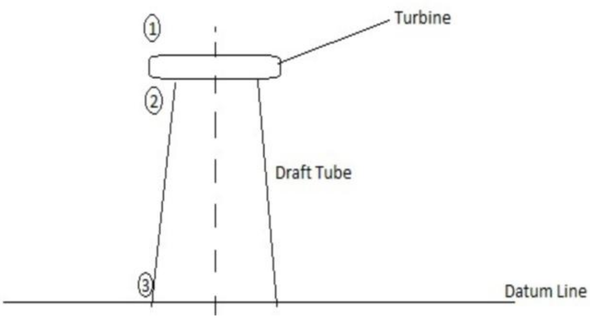


*fig: schematic representation of section of draft tube*

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

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

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 $19D_3$. The rate of change in recovery due to cone angle is more at small length of draft tube. The efficiency variations indicated that efficiency had gradually increased as the $L/D_3$ ratio increased up to 19 but after that effective increase of efficiency was very less due to small increase in recovery so increasing the length beyond $19D_3$ would not prove economical and also lead to problem of cavitation in turbine. The most of the hydro power plants have used straight conical draft tube around $19D_3$, diffuser angle 3.6° to 6° and hence the results from numerical simulation were validated. [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 using ten nodes tetrahedral element for good meshing on curvature parts. 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. The results were found in good agreement with each other. [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



ISSN: 2278 – 7798 

initial analysis. The study was carried out for four height $h/D_1$ and five lengths $L/D_1$ ratio in Design Modeller. Shear Stress Transport (SST) turbulence model in ANSYS CFX code was used for analysis due to boundary curvature in elbow and diffusing flow. It was observed that due to insufficient length for conversion of kinetic energy at low height ration and large eddy formation and flow separation, the efficiency of draft tube for constant mass flow rate of 20000 kg/sec has parabolic variation giving maximum efficiency at height ratio of 2.24. 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. The best performance was achieved at height ratio of 2.24 and length ratio $L/D_1$ of 6.0 and the results were validated with experimental data available. [4]

Vishal Soni, Amit Roghelia, Jaymin Desai, Vishal Chauhan worked on draft tube for high head Francis turbine. The turbine design initiated by CFD at Jyoti Ltd., project 'Swayam' was optimised. This initial design of draft tube was simulated by CFD at BEP and large amount of swirl was found at downstream of runner which resulted in more hydraulic losses and eventually decrease in the magnitude of static pressure recovery. 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 this study, the turbulent fluid flow in the Holleforsen hydropower draft tube model was simulated with the commercial fluid flow code Fluent 6.2 and the turbine model was mounted in VAUB`S turbine rig at the Alvkarleby Laboratory in Sweden. First experimental measured database conducted at 60% load of Kaplan turbine to obtain near optimal point of operation which is close to the best efficiency. 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. Using a model of bi-dimensional turbulence, isotropic and stationary they calculated the velocity and pressure fields in draft tube, and velocity profiles at five sections of draft tube. the Navier-Stokes equation was adopted for K-ε model for turbulence and SIMPLE algorithm for velocity-pressure. Standard wall functions were used to take the effects of friction near the wall. The numerical results were observed to have low difference with the experimental data because of the wall effects generated inside the draft tube that simulation does not take in account. 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. CONCLUSION

In this paper, various techniques are discussed about analysis of flow through draft tube. This study also discussed the parameters which affects the performance of draft tube and substantially the turbine and the power plant. Through the reviews it is found that the CFD can be used as a tool for analyzing the performance and the flow pattern inside the draft tube.

## REFERENCES

[1] Fluid Mechanics Fundamental and Applications, Second Edition, Yunus A. Cengel & John M. Cimbala, Tata Mc GrawHill Publication, Page 816.

[2] 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 (IJAEST), ISSN: 2249-913x, Vol. 2 No. 1 Mar-May 2012.

[3] 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.

[4] Vishnu Prasad, Ruchi Khare, Abhas Chincholikar, "Numerical Simulation for Performance of Elbow Draft Tube at Different Geometric Configurations"

[5] 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.

[6] 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.

[7] 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.

[8] 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.

[9] 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



ISSN: 2278 – 7798 

Systems, IOP Conf. Series: Earth and Environmental Science 12(2010) 012084, DOI: 10.1088/1755-1315/12/1/012084.

[10] 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: I402-I544.

[1]**Sumeet J. Wadibhasme**, B.E. Mechanical (Student), Priyaradshini Bhagwati College of Engineering, Nagpur University, Maharashtra (India).

[2]**Shubham Peshne**, B.E. Mechanical (Student), Priyaradshini Bhagwati College of Engineering, Nagpur University, Maharashtra (India).

[3]**Pravin Barapatre**, B.E. Mechanical (Student), Priyaradshini Bhagwati College of Engineering, Nagpur University, Maharashtra (India).

[4]**Santosh Barade**, B.E. Mechanical (Student), Priyaradshini Bhagwati College of Engineering, Nagpur University, Maharashtra (India).

[5]**Saurabh Dangore**, B.E. Mechanical (Student), Priyaradshini Bhagwati College of Engineering, Nagpur University, Maharashtra (India).

[6]**Shubham Harde**, B.E. Mechanical (Student), Priyaradshini Bhagwati College of Engineering, Nagpur University, Maharashtra (India).

[7]**Prof. Shailendra Daf**, Asst. Professor, Department of Mechanical Engineering, Priyaradshini Bhagwati College of Engineering, Nagpur University, Maharashtra (India).



ISSN: 2278 – 7798 