Proceedings of 8th IOE Graduate Conference
Peer Reviewed
ISSN: 2350-8914 (Online), 2350-8906 (Print)
Year: 2020 Month: June Volume: 8

Design & Study of Conical Draft Tube For Francis Turbine For Steady Flow

Bhuwan Karki [a], Subodh Kumar Ghimire [b], Hari Bahadur Dura [c]

[a, b] *Department of Automobile and Mechanical Engineering, IoE Thapathali Campus, Tribhuvan University*
[c] *Department of Mechanical and Aerospace Engineering, IoE Pulchowk Campus, Tribhuvan University*
Corresponding Email: [a] bhuwan.karki7@gmail.com, [b] subodh@tcioe.edu.np, [c] duraharis@pcampus.edu.np

Abstract

Draft tube helps to recover the pressure head by utilizing the kinetic energy of water flowing through reaction turbine. Different geometries of draft tube have been designed by changing the diffuser angle and output cross-section. The numerical analysis have been carried out in ANSYS16 CFX and optimum values of length and diffuser angle are found for the maximum efficiency and head recovery. The divergence angle are varied from 4° to 9.5°. Length to Inlet diameter ratio (L/D) of 13 has been chosen for flow analysis to check maximum Pressure recovery that needs to be optimized. The ratio of L/D lies in between 10 to 20 for best pressure recovery. Ultimately for the model which has shown back flow, the CFD simulation will be terminated. It has been found that model with diffuser angle 5° has maximum pressure recovery. Significant improvements has been done and validated with the mathematical model. Performance has been enhanced in this research work by optimizing the geometry.

Keywords

Draft tube, Pressure recovery, Losses, Efficiency

1. Introduction

The efficiency of hydraulic reaction turbine is enhanced by the draft tube [1]. In reaction turbines such as Francis and Kalpan turbines the diffuser tube is installed at the exit of turbine known as draft tube. The draft tube at the end of turbine increase a pressure to a higher extent without fear of back flow from the tailrace [2]. The simplest design of draft tube is a pipe of gradually increasing cross section area i.e. conical draft tube. One end of the draft tube is connected to the outlet of the runner while the other end is submerged below the level of water in the tail race [3]. The draft tube has following two purposes. Firstly, it permits a negative head to established at the outlet of the runner thereby increase the net head on turbine. Secondly, it converts a large proportion of the kinetic energy rejected at the outlet of the turbine into useful pressure energy. The head recovery is higher at smaller at lower L/D ratio while negligible improvement after L/D ratio greater than 19. At L/D ratio above 19, the turbine is susceptible to cavitation as well as the draft tube is not economically feasible [4].

Similarly, Umashankhar nema et.al [5] designed a draft tube of length (22.5m, 30.4m, and 38.4m with a diffuser angle of 4^0. The performance of all three draft tube had been evaluated using ANSYS CFD, the inlet velocity of water is taken as 10.01m/s. The length of 30.4m with diffuser angle of 4^0 is found to be optimum. The vishwanath et.al [6] studied the three draft tube shapes by varying the outlet cross section of the tube. The researcher designed (inlet-outlet) cross section area as: square-ellipse, square-circle, square-square. The value of L/D ratio is 2.4 and 5, diffuser angle of 4^0, 5^0 and 6^0. The paper finds the best pressure recovery for square to ellipse cross sectional area. It was also noted that the increase in length showed significant improvement in draft tube efficiency.

Jitendra [3] studied various type of elbow type draft tube with diffuser angles: 0^0, 10^0, 20^0 and 30^0. The CFD simulation is used to reduce the cost of prototyping. Navier-Stokes flow analysis for hydraulic turbine draft tubes is carried out by W. Shyy et.al [7]. The three-dimensional turbulent viscous flow analyses for hydraulic turbine elbow draft tubes are performed by solving Reynolds averaged

Navier-Stokes equations closed with a two-equation turbulence model. A statistical method for pressure pulsation is used to determine the pressure of the draft tube. Jing Yang et. al [8] studied the effect of inlet cavitation on swirling flow in draft-tube cone. The study found strong correlation between the swirling flow and cavitation. The swirling flow was not only affected by the load but also significantly influence the cavitation development.

The draft tube is designed and CFD simulation [9] is done in order to study the effect of diffuser angle, L/D ratio, outlet cross-section (i.e. ellipse) and swirl velocity. Similarly for the elliptical outlet cross section, the ratio of major axis and minor axis is varied keeping the outlet area constant in order to find optimal radius ratio. The effect of swirl (i.e axial and radial) on various elliptical shaped outlet is studied. The baseline model has the diffuser angle of 5° and L/D ratio of 13.

2. Methodology

2.1 1-D calculation

For the one dimensional calculation, the flow of water is based in the principal of Bernoulli equations given by the

$$\frac{p}{\rho g} + \frac{v^2}{2g} + z = Constant \qquad (1)$$

along stream line head recovery is obtained

$$H = \frac{\frac{v_1^2 - v_2^2}{2g} - 0.25\frac{v_2^2}{2g}}{\frac{v_1^2 - V_2^2}{2g}} \qquad (2)$$

Head loss with out friction is calculated as

$$Headloss = \frac{Tp_1 - Tp_2}{\rho g} \qquad (3)$$

Equation 2 gives the recovery head for the one dimensional flow of the fluid in the draft tube. Where v_1 is the inlet axial velocity in the draft tube and v_2 is the axial velocity at the outlet of the draft tube. Similarly $0.25\frac{v_2^2}{2g}$ is the head loss in the draft tube. Equation 3 gives the head loss without friction, Tp_1 is total inlet pressure and Tp_2 is total pressure at the outlet of the draft tube.

2.2 CFD analysis

2.2.1 Geometry

The geometry of draft tube is modeled in solid works software. The length of pipe for this study is is 10 m and with inlet diameter of 0.769m. The L/D ratio of the baseline model is 13. The figure 1 shows the schematic diagram diffuser angle model at 5^0.

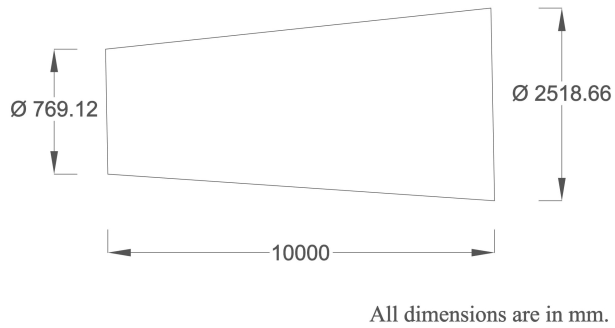

Figure 1: Conical draft tube for 5^0

2.2.2 Meshing

Automated unstructured tetra mesh is generated in Ansys CFX. Figure 2 shows the mesh generated in meshing module of Ansys CFX. The mesh given is for the simple draft tube with circular inlet and circular outlet.

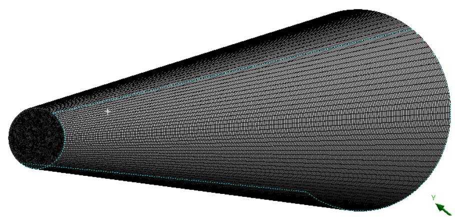

Figure 2: Mesh generated in Ansys Software

2.2.3 Physics setup

The inlet boundary condition is specified as a velocity inlet with the flow condition duplicating the runner outlet. The outlet boundary condition is specified as atmospheric condition i.e. gauge pressure of 0 atm. The inlet condition is taken with inlet velocity condition with magnitude of 10.1m/s. Various conditions with axial and circumferential velocity conditions are taken into account during the simulation. The details of the geometry is given in figure 1. The wall of draft tube is considered as no slip wall. No-slip conditions zero relative velocity between the first fluid layer and wall. The body force on the fluid is acceleration due to gravity. The data for the physics setup is summarized in the table 1.

S.N	Name	Boundary Type	Values
1	Inlet	Velocity	10.1 m/s
2	Outlet	Avg. static pressure	0 Pa
3	Wall	No slip wall	NA

Table 1: Boundary conditions

2.3 Mesh independence study

The mesh elements size up to 5000, the pressure recovery is varied. The simulation is done with mesh elements number 914000. The mesh independence study is shown in figure 3.

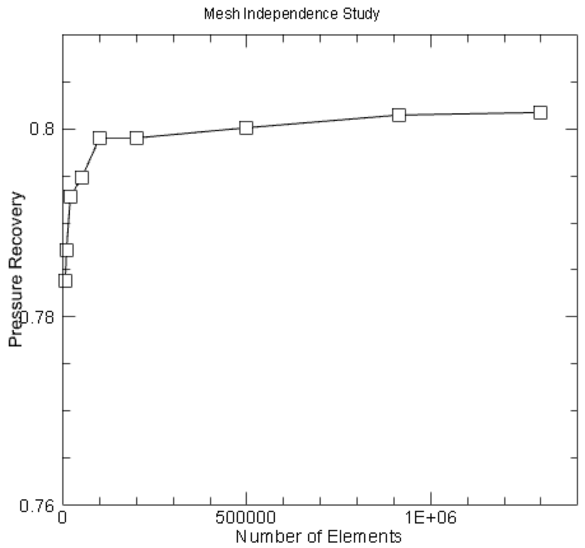

Figure 3: Mesh independence study

2.4 Case studies

In order th systematically study the effect of various parameters in the design of conical draft tube, following studies are carried out.

2.4.1 Variation of diffuser angle

For a given inlet diameter and constant length of draft tube are studied with the variation of diffuser angle ranging from 4^0 to 9.5^0. The variation of diffuser angle for different model is made with the variation of 0.5^0. The study will enhance how the diffuser angle affects on the performance of the draft tube. The study will includes the pressure recover at different model at constant L/D ratio.

2.4.2 Variation of L/D ratio

The L/D ratio is one of the important parameter to determine the efficiency of draft tube. For a given diffuser angle, different model with the variation of L/D ratio is focused. The variation of L/D is taken as 1, 2, 4,8, 10, 20, 30. The different model of different L/D ratio is generated for the simulation to study the head losses and efficiency of draft tube. The study is based on constant inlet diameter for different L/D ratios.

2.4.3 Variation of outlet Shape (ellipse)

The draft tube is modified to increase the pressure recovery. So that best model of different diffuser angel is modified to elliptical shape keeping the area constant equal to the outlet area of optimum model of diffuser angle. The major and minor axis is changed for the different elliptical shape and study is carried for the best recovery of the pressure. The ratio of major axis/minor axis is considered as 1,1.2, 1.4, 1.6 up to 3 for 11 model for the CFD simulation.

3. Result and Discussion

3.1 Validation

The Simulation result is verified with the experimental data of draft tube of hydroelectric stations [10]. The total head loss is calculated for the different value of L/D ratio. The result shows that beyond L/D=20, the head loss increase with less efficiency of draft tube. Figure 4 shows comparison of CFD data with the experimental results. The CFD simulation at the ratio of 13 has the minimum total head loss with the deviation of 4.48 percentage.

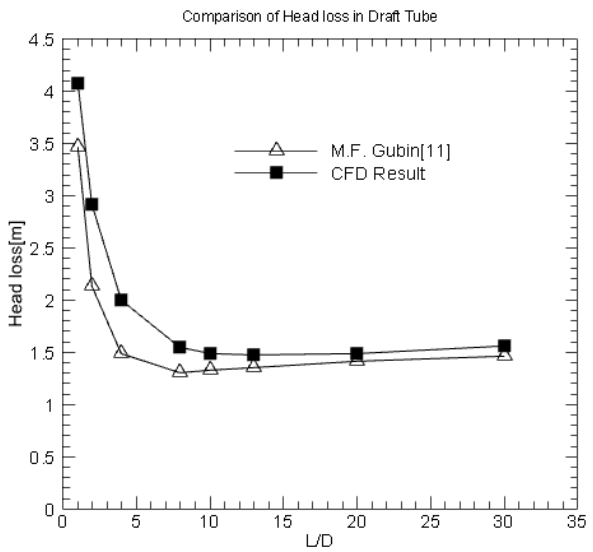

Figure 4: Comparison of total head loss for different L/D ratios

3.2 Effect of diffuser angle

Draft tube diffuser angle are varied with in the limit of 4° to 9.5°. Pressure variation with the increment of the diffuser angle is shown in the figure 5. The pressure is varied from 4° to 9.5°. Is found that conversion factor of velocity to the pressure at the outlet of the draft tube is more at five degree. From figure 3, it is found that pressure recovery first increase up to 5° and it decreases constantly up to 9.5°. The best model for the prototype is at diffuser angle at five degree(5°). With the increase of the diffuser angle cavitation is more likely to occur.

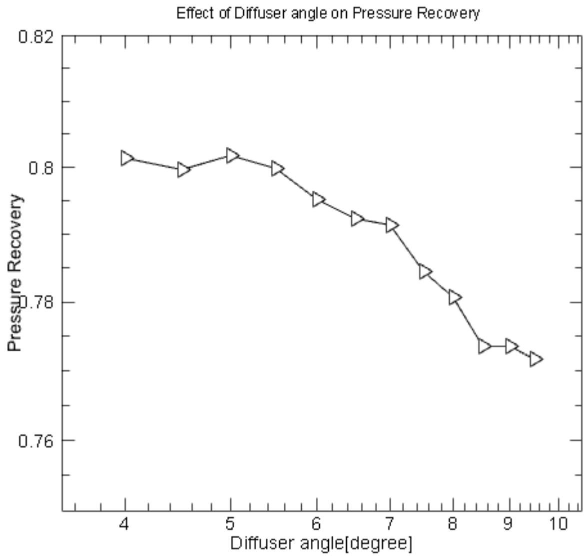

Figure 5: Pressure recovery in different diffuser angle

3.2.1 3D simulation of optimum model

The 3D simulation shows that pressure recovery for the simple conical draft tube of diffuser angle 5^0. The pressure distribution along the mid-plane of the simple draft tube is shown in figure 6.

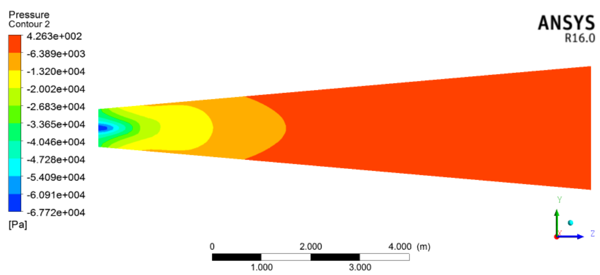

Figure 6: Pressure contour at 5° model

3.3 Effect of L/D ratio

For a given diffuser angle, the pressure recovery will increase with the increment of L/D Ratio. The analysis

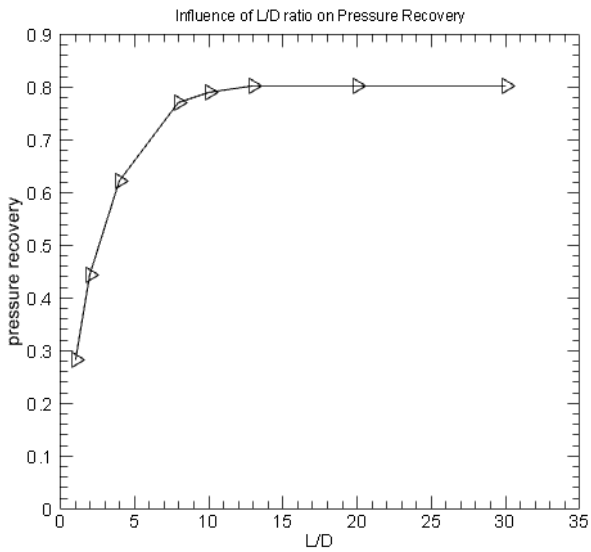

Figure 7: Pressure recovery at different L/D ratio with constant inlet and outlet areas

shows that the sharp increment of pressure recovery when L/D is 1, 2, 4, 8, 10. Beyond the 20, increment doesn't enhance the pressure recovery of the draft tube. With the increment of L/D ratio, it will enhance the friction loss which reduces the pressure recovery of the draft tube. The figure 7 shows the plot between pressure recovery and ratio of L/D.

3.4 Effect of outlet shape (ellipse)

3.4.1 Influence of R_{major}/R_{minor}

The ratio of major axis and minor axis is changed to get the different model of the draft tube. The pressure recovery is higher in elliptical draft tube rather than simplest design of conical draft tube. The increment of pressure recovery in elliptical outlet draft tube increases significantly when the Ratio of R_{major}/R_{minor} reaches to 2.4. Beyond the 2 the pressure increases almost constantly when the ratio reaches to 3 is shown in figure 8. The radial velocity is varied such as 1m/s, 4m/s and 6m/s. The increment of radial velocity prone occur the rotational flow from the draft tube, which causes disturbances as well as vibrations leading the less recovery of draft tube. The total inlet velocity is assume to be constant during the study of effect radial velocity. The pressure recovery is decrease and head loss increase with the increment of radial velocity which is uneconomical to the draft tube.

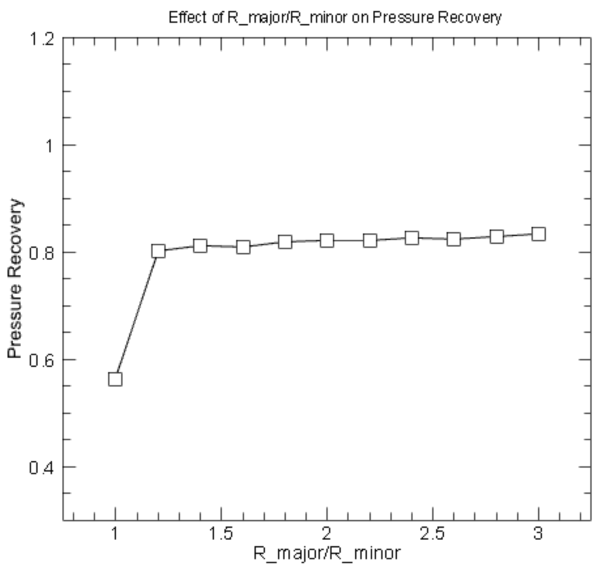

Figure 8: Influence of R_{major}/R_{minor} on pressure recovery

3.4.2 3D Simulation of elliptical model

The pressure variation along the mid-plane of the circular inlet to elliptical outlet is given in figure 9. The optimal radius ratio of the elliptical outlet is 2.4.

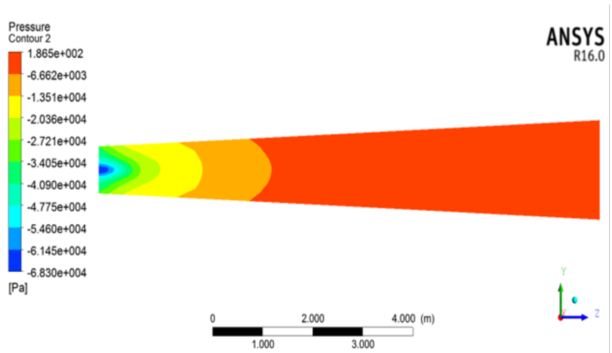

Figure 9: Pressure contour on the draft tube

3.5 Effect of radial velocities

The increase of radial velocity component leads to decrease in the pressure recovery. The radial velocity plays significant role in the performance of the draft tube. Figure 10 shows the influence of radial velocities on radius ratio of the elliptical outlet. It is observed that the pressure recovery is significantly influenced by the component of axial and radial velocities. Larger the radial velocity, the lower the efficiency.

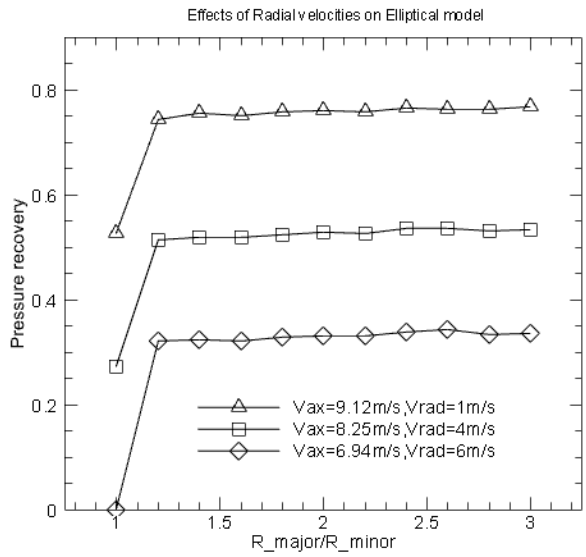

Figure 10: Effect of radial velocity at different elliptical outlet shapes

The influence of radial velocity on the diffuser angle is shown in figure 11. The radial velocity is decreased linearly in the diffuser angle.

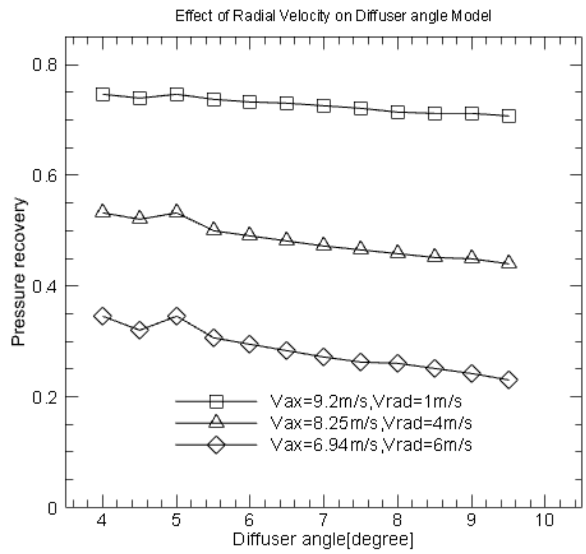

Figure 11: Effect of radial velocity at different diffuser angle

4. Conclusion

The current study explored the influence of various geometric and inlet condition on the performance of the simple conical type draft tube. Diffuser angle, outlet cross section (ellipse) & radial velocity component is varied and performance was evaluated using analytical method and numerical simulation. The maximum pressure recovery at diffuser angle 5°

and L/D ratio 13, is found to be 0.801751 with a efficiency of 82.34%. Further modified for the model in elliptical cross sectional having constant area at the outlet section enhanced the performance of the draft tube. The study found that there is inverse relation between radial velocity and performance parameters i.e. pressure recovery and efficiency. The minimal head loss is found to be 1.41 meter at diffuser angle of 5°. The pressure recovery of elliptical shape found to be 0.827542 for larger L/D ratio. The deviation of total head loss of the CFD simulation is found to be 4.48% compared with the experimental data found in literature.

Acknowledgments

The authors acknowledge Department of Automobile & Mechanical Engineering, Thapathali Campus, Department of Mechanical & Aerospace Engineering, Pulchowk Campus for providing continuous help and support to conduct this research.

References

[1] Ali Abbas and Arun Kumar. Development of draft tube in hydro-turbine. *A review,International Journal of Ambient Energy*, 2015.

[2] Amit Roghelia et al. Design development of optimum draft tube for high head francis turbine using cfd. *proceedings of the 37th international and 4th national conference on fluid mechanics and fluid power*, 2010.

[3] 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*, 2, 2017.

[4] Sumeet et al. , hydraulic turbine draft tube: Literature review. *International Journal of Science, Engineering and Technology Research (IJSETR)*, 5, 2016.

[5] Umashankar Nema and Rohit Rajvaidya. Design and evaluation of performance of conical type draft tube with variation in length to diameter ratio. *International Journal of Engineering Sciences and Research Technology.*, 2017.

[6] Vishwanath et al. Design and optimization of draft tube to increase the efficiency by varying the geometry of the draft tube outlet,. *International Journal for Research in Applied Science & Engineering Technology*, 5, 2017.

[7] W. Shyy et.al. *Navier-stokes flow analysis for hydraulic turbine draft tubes*. 1988.

[8] Jing Yang et. al. Effects of inlet cavitation on swirling flow in draft-tube cone. 2018.

[9] J McNabb et al. Cfd based draft tube hydraulic design optimization. *, IOP Conference Series: Earth and Environmental Science*, 2014.

[10] M. F. Gubin. Draft tube of hydro-electric stations. *Amerind Publishing Co. Pvt. Ltd*, 1973.