

Numerical and Experimental Analysis of Draft Tubes for Francis Turbine

Tarang Agarwal¹, Shreyash Chaudhary¹ and Shivank Verma¹

¹Invertis University, Bareilly – 243123, Uttar Pradesh, India;
tarang.agarwal1@gmail.com, shreyashchaudhary000@gmail.com, shivank.v@live.com

Abstract

Objectives: Draft tube is an important component of a hydro turbine (reaction) because it permits a negative head thereby increasing the total head on the turbine. It is located below the runner and allows deceleration of flow at exit. Since, the efficiency of the turbine can be increased by increasing the overall-efficiency of the draft tube, it's important to identify and optimize the design parameters which affect the efficiency of draft tube. **Methods/ Statistical Analysis:** In the present study, numerical and experimental analysis was done on different draft tubes in Francis turbine. In experimental study, different types of draft tubes were designed, fabricated and tested on a Francis turbine test rig. For conical draft tube, cone angle and cross section (outlet) was varied to understand the flow dynamics. Further, elbow draft tube (rectangular section) was also designed and tested to determine Best Efficiency Point (BEP). Numerical analysis was done using ANSYS CFX for various draft tubes which were designed in ANSYS Design Modeler. **Findings/Results:** The results of numerical analysis were validated experimentally. High pressure zones were identified in conical and elbow draft tube, although, overall pressure was less than atmospheric pressure. The efficiency of Francis turbine using 8 degree cone angle gives higher efficiency. For angles greater than 8 degree, backflow was observed. For variable discharge conditions (part-load), elbow draft tube can be used because of lesser variations in efficiency at different flow rates. **Applications/Improvements:** Further, conical draft tube with square section was also analyzed. Higher values of efficiency were obtained at higher speed of the turbine. Hence, it is advisable to further study this type of draft tube.

Keywords: CFD, Draft Tubes, Efficiency, Francis Turbine, Suction Pressure

1. Introduction

At present, a major section of electricity production is covered by the utilization of conventional energy resources. The demand of electricity is increasing rapidly which has paved the way of non-conventional and renewable energy sources to act as alternate energy resources. These alternative energy resources include – Hydro energy, Nuclear energy, Wind energy, solar energy, Bio fuel etc. Among all these sources of energy, hydropower is becoming more prominent because it is a clean and abundant energy resource. Moreover it also generates about 16.3% of the total electricity in world (3801.48 TWh in 2013)¹.

In order to increase this share, construction of new hydro-power plant and modification of the existing hydropower plants is required. Since the main part of a hydropower plant is the turbine, efforts are made to increase the efficiency of turbine. Even the small improvements in the efficiency of turbines are important in terms of power generated and economical aspects. In a turbine, the fluid flow is directed to the runner in a manner such that the energy losses prior to it are minimum. After the runner, the flow emerges with a mean velocity and kinetic energy which is a loss in terms of energy. If the runner is situated above the tail race, a part of potential energy equal to the height difference between runner and tail race is lost. This is true for all impulse turbines.

*Author for correspondence

In² suggested attaching a cylindrical tube after the runner of reaction turbines. One end of this tube was connected to the turbine and the other was situated below the tail race level. Originally these tubes were known as suction tubes. Later in 1960, they were renamed as draft tubes. From fluid flow perspective, draft tube is the most challenging component to design because of the interaction of complex flows. Draft tube serves the following two purposes:

1. Installation of the turbine runner above the tail water level (avoiding cavitation).
2. Conversion of kinetic energy at runner exit into pressure energy (reaction principle)

The third function is less obvious and up to now not clearly addressed is that the draft tube determines the tail water level number and tail water Froude and hence, the non-dissipative losses³.

The aim of this study is to analyze the effect of efficiency of a turbine by use of different types of draft tubes. More specifically, this study seeks to make a comparative attempt at assessing efficiencies of different types of draft tubes in order to determine the effect on the efficiency of Francis turbine and optimization of existing draft tubes.

2. Working Principle

The underlying principle can be understood by using Bernoulli's equation between section 1-1 and 2-2 as shown in Figure 1.

$$p_1/\rho g = p_{atm}/\rho g - (H_s + (V_1^2/2g) - (V_2^2/2g) - h_f) \quad (1)$$

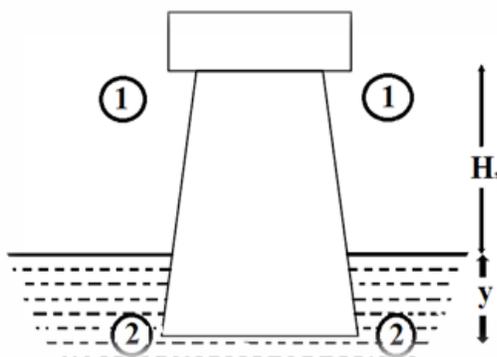


Figure 1. Conical draft tube.

Where p_1 is the pressure at exit of runner (inlet of draft tube), p_{atm} is atmospheric pressure, V_1 is the velocity at runner exit (draft tube inlet), V_2 is the outlet velocity (tail

race), H_s is the suction head or draft tube height, h_f are the frictional losses, ρ is fluid density and g is gravitational constant.

From eq. (1), it can be understood that draft tube generates negative pressure (vacuum) which is utilized inside the turbine (reactionary principle). Draft tube plays an important role converting the excess kinetic energy (velocity of fluid) into pressure energy which is utilized inside the turbine. Hence, draft tube efficacy plays an important role in enhancing the overall efficiency of turbine.

3. Types of Draft Tubes

According to the geometrical design draft tubes can broadly be classified into three types:

1. Conical type,
2. Curved or Elbow type, and
3. Bell Mouth type.

The conical (straight) draft tube Figure 1 is the simplest form of draft tube and it has excellent hydraulic characteristics. The conical draft tube consists of three parts: The initial part starting from the axis of the runner blades, the cone proper and the exhaust chamber. This type of draft tube is runner size of up to 2.5m (small and medium size runner).

Curved draft tube is the basic type of draft tube used in vertical hydraulic turbines. Curved draft tubes also have three different sections, namely initial cone, curved or elbow section and outflow diffuser. The initial part of tube is similar to that of a straight conical draft tube Figure 2. Curved draft tube is generally used in runner size of upto 10m (medium and large size runner). Further classification can be done on the basis of design of elbow or curved section.

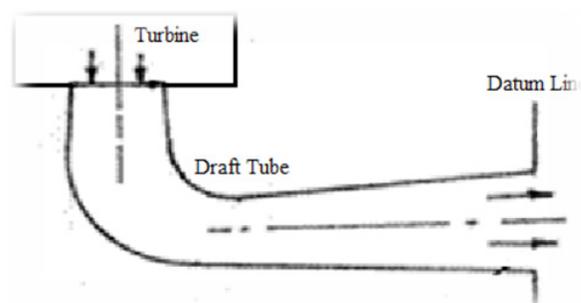


Figure 2. Curved draft tube.

The bell mouth spreading draft tube is a straight axis tube but differs from the conical draft tube in aspects of shape of its walls (on the basis of some non-linear law). The shape resembles a trumpet with sharp flare Figure 3 at the exit. This type of draft tube is generally used for runner size of up to 5 m (medium size).

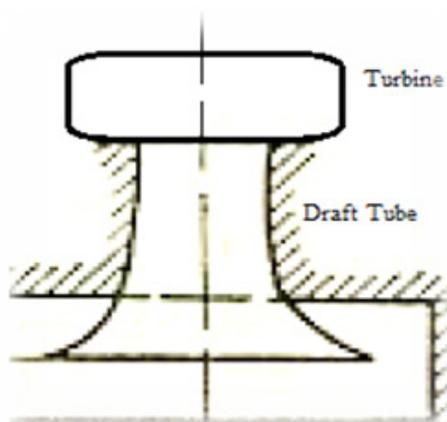


Figure 3. Bell mouth draft tube.

4. Literature Review

There have been various studies on the existing draft tube designs. In⁴ used CFD code for a 3D viscous and turbulent flow inside Francis turbine running at different speeds. It was concluded that whirl and meridional velocity components (outlet) depends on the operating regime of turbine and these velocities also affects the overall losses inside the draft tube. Moreover, variation in losses and efficiency of draft tube was found to be parabolic with respect to the speed factor. The Best Efficiency Point (BEP) was also found to be shifted towards higher speed factor. In a later study⁵, it was concluded that length and diffuser angle has a significant effect on the performance of straight conical draft tube In⁶ conducted a numerical analysis of a Francis turbine. It was concluded that the guide vane thickness was an important parameter and optimal thickness can lead to maximum efficiency value.

In⁷ analyzed elbow draft tube numerically and determined the parameters affecting the performance of draft tube. Height (above tail race) and length were identified as important parameters which influences the draft tube performance while variation in mass flow rate had very less effect on draft tube efficiency In⁸ used various designs of existing bend draft tube and analyzed them numerically. CFD was identified as an important tool in

identifying the important parameters, their dependence and understanding of working mechanism. Hence, CFD can be used to improve the design of existing draft tubes In⁹ conducted an experimental study for establishing a relationship between flow separation inside the draft tube and efficiency for a bulb turbine. It was concluded that the flow separation is 3D in nature and flow separation zone extends till far upstream along the corner of the draft tube In¹⁰ analyzed draft tube using a numerical simulation. The results of pressure and velocity at inlet and exit of draft tube obtained by CFD analysis were then compared with the results obtained by experimental procedure. Both results were in good agreement with each other. Thus it was observed that by using CFD analysis results one can easily predict almost nearer results to that of experimental analysis. It was also concluded that CFD analysis can be used as a tool for reducing the time and cost for a given investigation.

5. Problems in Draft Tubes

After an extensive study of literature available, following three were identified as problems in draft tube:

5.1 Vortex Rope or Swirling Flow

Vortex rope is one the main problem that occurs in draft tube. This vortex rope rotates and generates a rotating oscillatory pressure field. These pressure oscillations can lead to severe vibrations and other losses in the draft tube. In¹¹ concluded that vortex rope intensity depends on 3 parameters viz. specific speed of the runner, shape of the runner and operating regime. In^{12,13} carried out an extensive experimental study to identify various parameters involved in deceleration of swirling flow (instability). Figure 4 shows corkscrew vortex formation in the numerical simulation.



Figure 4. Corkscrew vortex in a model¹³.

Many methods have been developed to minimize the shedding of vortex rope without affecting the operational efficiency. In¹⁴ suggested that pressure and flow below the runner discharge area can be controlled by extending the runner cone which in result will control the swirl in the draft tube. He also suggested that attachment of various stabilizer fins inside the draft tube will be beneficial for mitigating the draft tube swirl and pressure fluctuations. Structures such as splitter plates etc. were also proposed as the solution to reduce swirl intensity inside the draft tube. However, these structures might create adverse effects at other operating regimes due to geometrical corrections¹⁵. In¹⁶ developed a technique of injecting small water flow rate (1-2% turbine discharge) inside the draft tube cone (forced excitation) which cancels the self-induced pressure fluctuations generated by the vortex rope at partial discharge. In¹⁷ concluded that breakdown of the helical vortex (at part load) is directly related to the severe deceleration of flow at the downstream (runner). Moreover, injecting of water from the cone tip was also suggested to minimize the instability inside the draft tube.

5.2 Surging

Draft-tube-surge is the term used for the pressure fluctuation observed in draft tubes. It is observed that when surging is strong it results in vibration of draft tube and penstock which ultimately leads to loss of power and may also cause rupturing of draft tube liner. At part load conditions, whirl frequency of the vortex rope equals to the frequency of hydraulic turbine (resonance) which leads to surging in draft tube¹⁸. This phenomenon also affects the limitations of hydro turbine due to restrictions in certain operating range¹⁹. One possible solution to minimize this is to supply certain amount of air inside the draft tube. Moreover, compressed air can also be supplied whenever necessary.

5.3 Cavitations

Another major problem that occurs in draft tube is cavitations. In reaction turbines when the discharge is reduced to a certain amount from its original discharge owing to rotation, cavity is formed at the inside the draft tube. This process of cavity formation is called cavitations Figure 5. The formation of cavity will lead to the vibration in the draft tube. Rate of discharge is an important parameter to reduce cavity as the radius of cavity generally increases with reduction in discharge¹⁹.



Figure 5. Damage in draft tube due to cavitation²⁰.

6. Design and Analysis

Design of draft tube requires deep understanding of hydrodynamics. The difficulty lies in the fact that the flow in the draft tube is influenced by many parameters such as operating condition of the turbine, type of the runner, geometry complexity of the tube etc. But on the basis of study of different experimental works we have tried to establish the dimension standards of the different types of turbines. The design of draft tube is based on design parameters from book by In². Figure 6 and 7 shows the basic design of conical and elbow draft tube. Other draft tubes were modified accordingly. Dimensions of designed straight conical draft tube are:

1. Throat Diameter (D_1) = 80 mm.
2. Cone Angle (2θ) = 16 degrees.
3. Length of conical part (L_{con}) = 600 mm.
4. Outlet Diameter (D_2) = 248 mm.

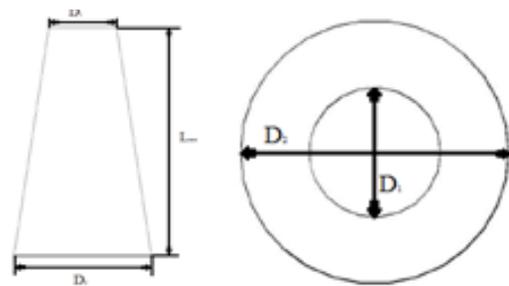


Figure 6. Designed straight conical draft tube.

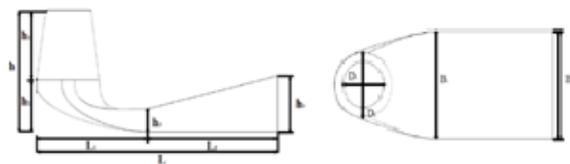


Figure 7. Designed curved draft tube.

Dimensions of designed curved draft tube are:

1. Throat Diameter (D_1) = 80 mm.
2. Cone Angle (θ) = 8 degree.
3. Height of Draft Tube (h) = 200 mm.
4. Length of Draft tube (L) = 380 mm.
5. Height of conical section (h_1) = 120 mm.
6. Diameter of inlet section of elbow (D_2) = 114 mm.
7. Height of elbow section (h_2) = 80 mm.
8. Length of elbow section (L_1) = 128 mm.
9. Width of the exit section of elbow (B_1) = 200 mm.
10. Height of exit section of elbow (h_3) = 35 mm.
11. Angle of elevation of diffuser (α_1) = 15 degrees.
12. Height of exit section of diffuser (h_4) = 91 mm.
13. Width of exit section of diffuser (B_2) = 200 mm.

6.1 Introduction to Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) is involves solving a fluid flow problem (or other related physical processes) using a numerical simulation. This methodology involves various stages such as geometric/mathematical modeling, discretization etc. Initially, geometric model is prepared in computer and mathematical model is identified (nature of problem). The problem is further discretized using techniques such as Finite Difference Method (FDM), Finite Volume Method (FVM) and Finite Element Method (FEM). Numerical solution means solving the identified numerical model by converting it into discrete algebraic problem (steady) or ordinary differential equation (unsteady). Hence, value of variables are calculated at each computational node and results are analyzed to understand the reality.

6.2 Modeling and Mesh Generation

The geometries were created in Design Modeler module of ANSYS 15. Figure 8 and 9 shows the conical draft tubes (8 degree cone angle and 3 degree cone angle) and elbow draft tube.

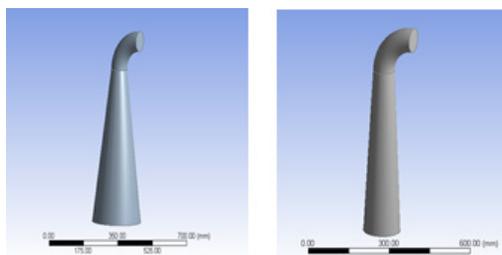


Figure 8. Conical draft tube with 8 degree and 3 degree cone angle.

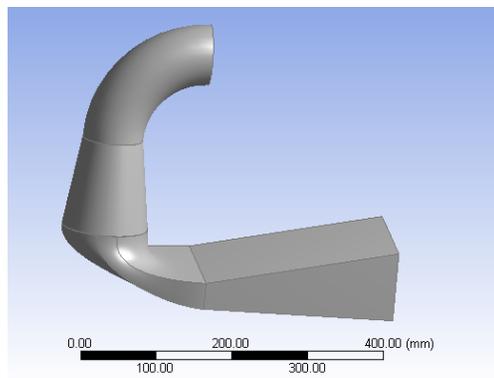


Figure 9. Model of elbow draft tube.

Mesh was created in MESH module of ANSYS. Figure 10 and 11 shows the meshed view of conical and elbow draft tube. Details of nodes and elements generated for each geometry are as below:

1. Number of Nodes: 45068, Number of elements: 142856 (Conical draft tube 8 degree).
2. Number of Nodes: 71012, Number of elements: 214203 (Conical draft tube).
3. Number of Nodes: 44765, Number of elements: 142323 (Conical draft tube 3 degree).

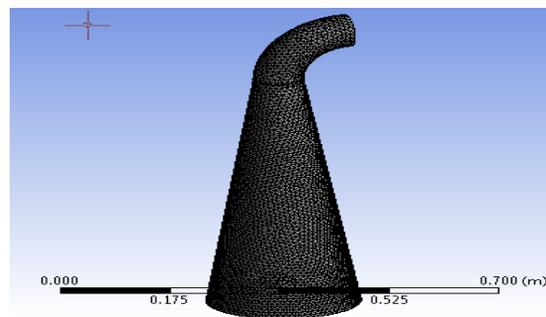


Figure 10. View of meshed conical draft tube.

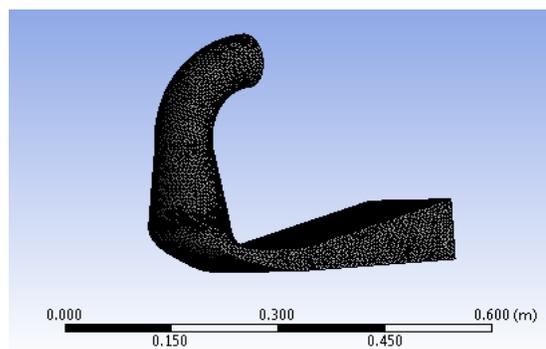


Figure 11. View of meshed elbow draft tube.

These meshes were generated again with finer and coarse grid options to determine grid independence. Variations in pressure and velocity were noted. Since, the variations were in negligible (order 10^{-3}), grid independence was tested.

6.3 Pre-Processing

ANSYS CFX was used for pre-processing in which domain, materials and boundary conditions were defined. Initially, water was selected as working fluid and reference was taken as 1 atmospheric pressure. Turbulence plays an important role inside the draft tube. Hence, Shear Stress Transport (SST) model was defined to analyze the flow inside the draft tube. Inlet and Outlet of draft tubes were created. Wall type boundary was defined on the outer section while inside domain was specified as fluid (water). The mass flow rate (as in experiment) was defined at the inlet and outlet pressure was kept uniform. Flow direction was kept normal to the boundary at inlet and no slip wall (smooth) condition was specified for wall boundary. Figure 12 and 13 shows the boundary conditions at inlet and outlet of conical and elbow type draft tube.

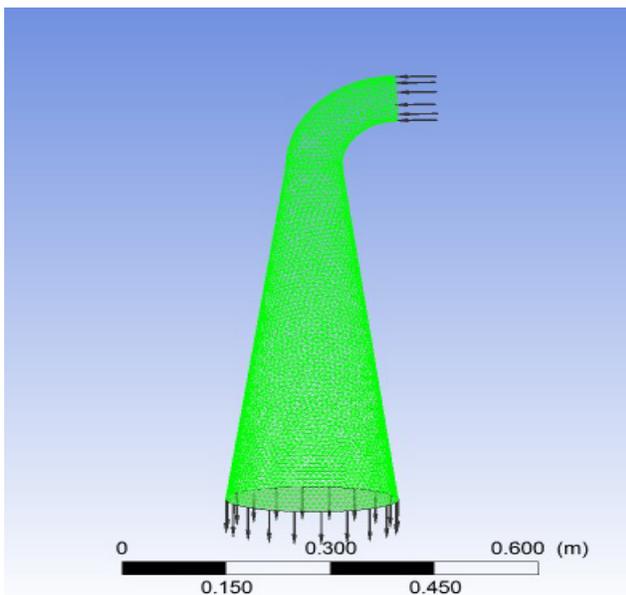


Figure 12. Inlet and outlet boundary condition for conical draft tube.

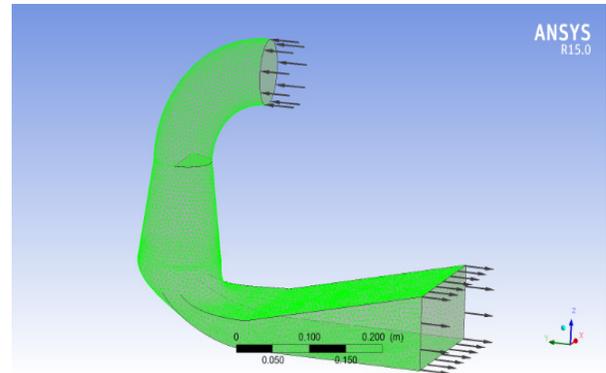


Figure 13. Boundary conditions for elbow draft tube.

6.4 Solution and Post-Processing

High resolution advection scheme with first order turbulence numeric was used. Auto timescale control and RMS criteria was specified. Maximum iterations were increased accordingly. The results of the simulation have been discussed in detail in section 5.

7. Experimental Setup

The experimental test rig was designed to obtain the efficiency of the system at different flow conditions and using different draft tubes. Test rig consists of a Francis turbine, draft tube, pump etc (Figure 14).

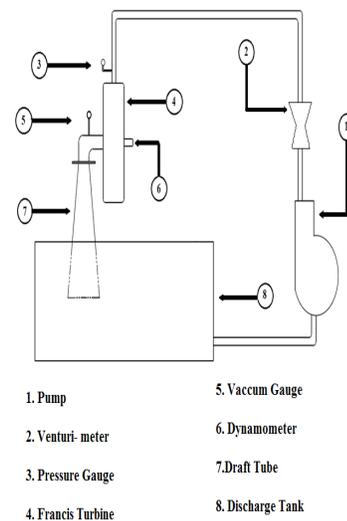


Figure 14. Schematic diagram of Francis turbine test rig.

Various draft tubes were designed and fabricated in the lab to study the flow pattern and their contribution to Francis turbine efficiency. Different draft tubes that were fabricated are as follows:

1. Straight conical draft tube (3° cone angle).
2. Straight conical draft tube (8° cone angle).
3. Straight conical draft tube with square section at outlet.
4. Curved elbow draft tube.

The fabricated draft tubes were then fitted on the system and efficiency of Francis turbine was calculated. Figure 15-17 show conical, conical with square outlet and elbow draft tube.



Figure 15. Conical draft tube having 8 degree cone angle.



Figure 16. Straight conical draft tube having variation in outlet section.



Figure 17. Side view and top view of curved elbow draft tube.

7.1 Procedure

Fabricated draft tubes were assembled with the turbine and efficiency was calculated by varying the flow conditions and draft tube itself. Since synchronous speed plays important roles in electricity generation, mostly those flow rates were selected which gave turbine speed close to synchronizing speed. System was switched on and was kept undisturbed for 5 minutes so that it may attain a steady state. Flow rate was controlled by using a flow regulating mechanism. The speed (revolution per minute) of the flywheel was measured by using a digital tachometer. This flywheel was coupled with turbine and rotates with the same speed as of the turbine. Once the flow rate was set, then the readings on the venturimeter, pressure gauge (gauge) and vacuum gauge were noted. Dead weight was placed on the hanger of the dynamometer attached to the turbine until the flywheel of dynamometer stopped. The reading on the spring balance was noted. Total dead weight to stop the flywheel of dynamometer was also noted. To calculate the output power, torque was calculated and multiplied with angular speed of the flywheel and input power was calculated using the conventional formula i.e. $\text{power} = \text{density (water)} \times \text{acceleration due to gravity} \times \text{head available at inlet} \times \text{discharge in cumecs}$. Results of experimental analysis have been discussed in detail in section 5.

8. Results and Discussion

8.1 Numerical Results

The pressure contours for conical and elbow draft tubes were plotted for specific cases. For conical draft tube with 8 degree cone angle, pressure increased from -10798.7 Pa to -610.2 Pa

(conservative). This variation in pressure shows conversion of kinetic energy into pressure, thereby increasing the efficiency of turbine. Velocity of water decreased from 2.65 m/s at inlet to 0.54 m/s at outlet. In theory, pressure and velocity both must decrease in draft tube but since we haven't connected the turbine (numerical analysis), efficiency of draft tube is estimated as a measure of overall losses. Hence, each computed value was substituted in Bernoulli equation and overall losses were calculated. From Figure 18 and 19, it can be seen that there is a large variation in pressure inside the bend section. Suction is created on the inner side (concave) while high pressure Figure 20 exists on the outer side (convex) of the bent section. Since overall suction occurs, velocity at inlet is higher than theoretical velocity (discharge/area of cross section). For conical draft tube with 3 degree cone angle, similar pressure contours were obtained. In both cases, the suction pressure obtained at inlet was same as in experiment. In case of 8 degree conical draft tube, suction pressure at inlet was 10798.7 Pa (numerically) and 11030 Pa (experimentally). Since the error in these values is around $\pm 2\%$ (within acceptable limits), model is validated.

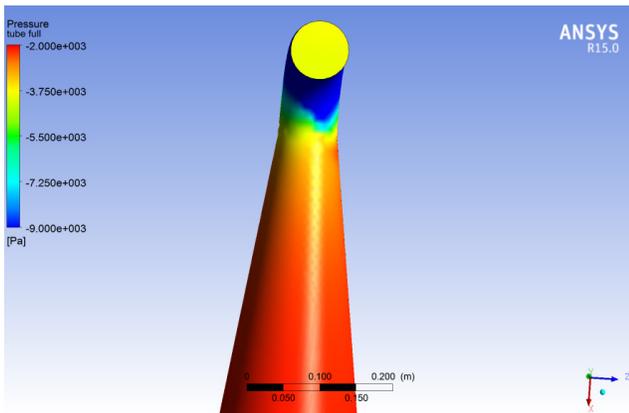


Figure 18. Pressure contours on conical draft tube.

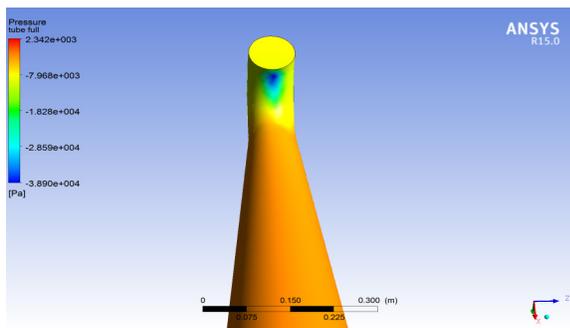


Figure 19. Pressure contours on inner edge of conical draft tube.

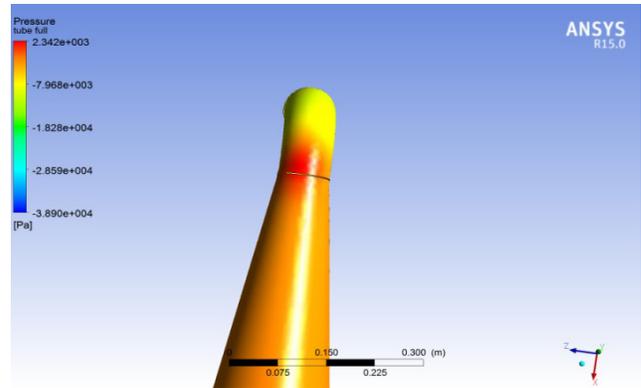


Figure 20. Pressure contours on the outer side of conical draft tube.

Figure 21 and 22 shows the pressure variations in elbow draft tube. Pressure at inlet was -9839.7 Pa and -797.4 Pa at outlet. Velocity decreased from 1.74 m/s at inlet to 0.46 m/s at outlet.

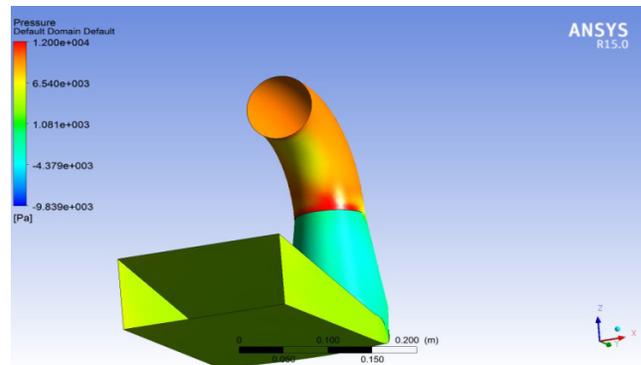


Figure 21. Pressure contours on concave side of elbow draft tube.

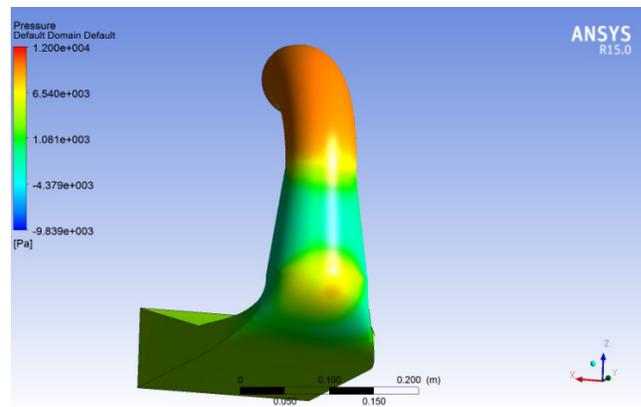


Figure 22. Pressure contour on convex side of elbow draft tube.

Head loss (h_f) was calculated in each case using Bernoulli equation. These losses are a direct measure of draft tube efficiency i.e. higher head loss means lower draft tube efficiency. The suction head (height of draft tube) is taken as 0.72m. Overall head loss in 8 degree conical draft tube (at 7.35 kg/s discharge) was 0.024 m of water. Similarly, losses in case of 3 degree conical draft tube (at 6.92 kg/s) was 0.041 m of water and with elbow draft tube was 0.059 m of water.

8.2 Experimental Results

Efficiency of Francis turbine was calculated using the experimental readings. Table 1 shows various parameters that were evaluated with conical draft tube (cone angle as 8 degrees). These parameters were also calculated for other flow rates (at specific rpm). Figure 23 shows the variation in efficiency with rpm of the turbine (at different mass flow rates). It is seen that maximum efficiency occurs at 1500 rpm of turbine when mass flow rate is 7.35 kg/s.

Table 1. Different parameters of Francis turbine (conical draft tube with cone angle 8 degrees).

Discharge Pressure (P_d)	0.31 kg/cm ²	Suction Pressure (P_s)	0.11 kg/cm ²
H ₁	0.5 cm	H ₂	1.4 cm
Dead Weight	2.0 kg	Spring Weight	0.7 kg
Head	4.1 m of water	Velocity Inlet (theoretical)	1.47 m/s
Mass Flow Rate	7.35 kg/s	Torque	1.59 Nm
Power In	0.295 kW	Power Out	0.2494 kW
Efficiency	84.39 %		

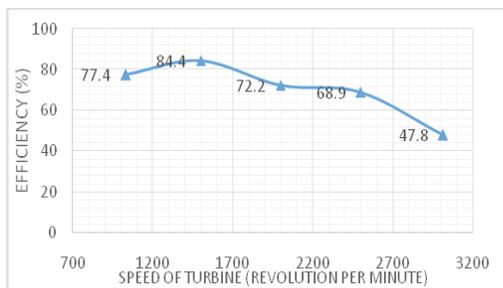


Figure 23. Variation in efficiency with rpm of turbine (conical 8 degree draft tube).

Same parameters were also calculated for Francis turbine with draft tube having 3 degree cone angle. Table 2

shows various parameters that were evaluated with conical draft tube with cone angle as 3 degrees. It can be seen from Figure 24 that the maximum efficiency occurs at mass flow rate 6.92 kg/s.

Table 2. Different parameters of Francis turbine (conical draft tube with cone angle 3 degrees).

Discharge Pressure (Pd)	0.34 kg/cm ²	Suction Pressure (Ps)	0.10 kg/cm ²
H1	0.5 cm	H2	1.3 cm
Dead Weight	1.5 kg	Spring Weight	0.7 kg
Head	4.4 m of water	Velocity (theoretical)	1.37 m/s
Mass Flow Rate	6.92 kg/s	Torque	1.59 Nm
Power In	0.31 kW	Power Out	0.24 kW
Efficiency	77.4 %		

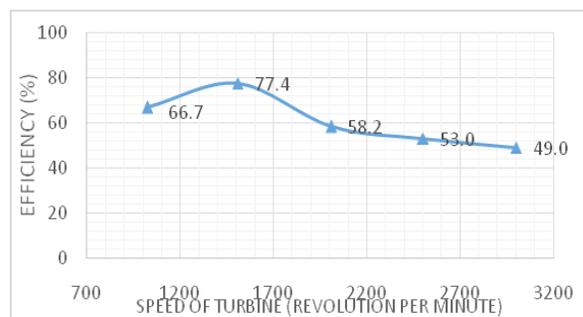


Figure 24. Variation in efficiency with rpm of turbine (conical 3 degree draft tube).

Table 3 shows various parameters that were evaluated with conical draft tube with square outlet. It can be seen from Figure 25 that the maximum efficiency occurs at 11.75 kg/s discharge condition. This flow rate is more than that in conical draft tubes. This trend is quite different as the value efficiency kept increasing with discharge (or rpm of system). Hence, it is advisable to further test this draft tube on setups which can accommodate higher discharge. Table 4 shows the parameters corresponding to elbow draft tube. It can be seen from Figure 26 that efficiency reaches as high as 81% but at higher flow rates (9.48 kg/s) as compared to rated discharge condition.

Table 3. Different parameters of Francis turbine (conical draft tube with square section).

Discharge Pressure (P_d)	0.33 kg/cm ²	Suction Pressure (P_s)	0.92 kg/cm ²
H_1	1.2 cm	H_2	1.8 cm
Dead Weight	1.5 kg	Spring Weight	0.7 kg
Generated Head	4.3 m of water	Velocity (Theoretical)	1.19 m/s
Mass Flow Rate	5.99 kg/s	Torque	1.0491 Nm
Power In	0.2530 kW	Power Out	0.1669 kW
Efficiency	65.95%		

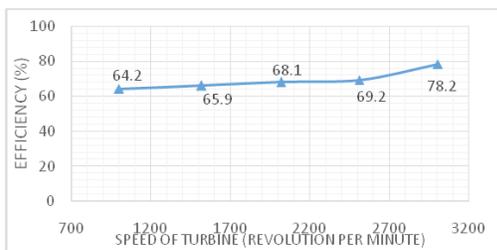


Figure 25. Variation in efficiency with rpm of turbine (conical draft tube with square outlet).

Table 4. Different parameters of Francis turbine (curved elbow draft tube).

Discharge Pressure (P_d)	0.325 kg/cm ²	Suction Pressure (P_s)	0.10 kg/cm ²
H_1	0.8 cm	H_2	2.0 cm
Dead Weight	2.0 kg	Spring Weight	0.7 kg
Generated Head	4.25 m of water	Velocity (theoretical)	1.68 m/s
Mass Flow Rate	8.4843 kg/s	Torque	1.5886 Nm
Power In	0.3537 kW	Power Out	0.2500 kW
Efficiency	70.69 %		

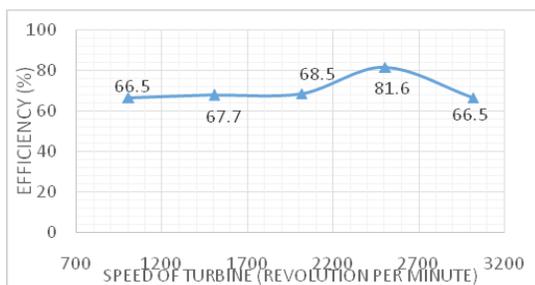


Figure 26. Variation in efficiency with rpm of turbine (elbow draft tube).

It is seen from Figure 27 that maximum efficiency of Francis turbine reaches as high as 84.4% when conical draft tube with 8 degree cone angle is used. The efficiency trend for 3 degree conical tube is same but the values are lower. For conical draft tube with square head, efficiency increases with discharge (and speed) and reaches as high as 78.2 % for discharge as high as 11.7 kg/s (rated is 7.4 kg/s). System may attain a higher efficiency at higher discharge but system limitations didn't allow us to go beyond this limit. For elbow type draft tube, efficiency of the system reached as high as 81.6 % but at a higher flow rate value i.e. 8.49 kg/s.

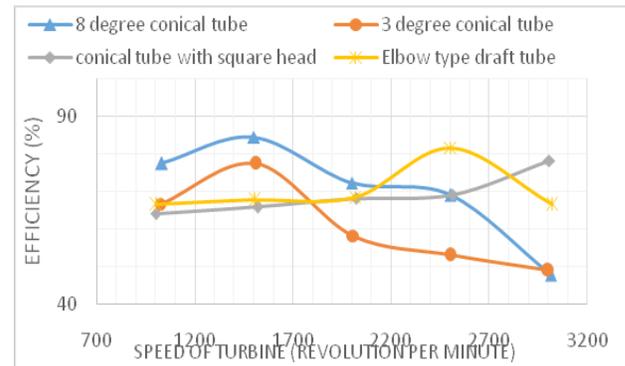


Figure 27. Comparison of efficiencies of Francis turbine using different draft tubes.

9. Conclusion

In the present study, numerical and experiment investigations were carried out on different draft tubes in Francis turbine at different flow conditions. Different types of draft tubes were designed, assembled and tested at different turbine speed and discharge conditions. Results obtained in Numerical analysis were compared with experimental results. The order of error in values was found to be within acceptable limits.

Although, the pressure inside the draft tube is less than atmospheric pressure, still high pressure zones were identified in conical and elbow draft tubes. These areas will be severely damaged when particles (sand) inside the water will impact against the surface. Other possibility of damage will be due to collapse of high pressure bubble (cavitations erosion). One way to avoid high pressure zones is by using baffle plates inside the draft tubes. This will also minimize the turbulence losses up to a certain extent. Further, it can be concluded Francis turbine has maximum efficiency (84.4%) when conical draft tube with 8 degree cone angle

is used. The efficiency for 3 degree conical tube is comparatively less. Hence, it is suggested to keep the cone angle in the range of 7-8 degree. Angles greater than 8 degree may also result in lesser efficiency due to backflow problem. Conical draft tube with square outlet gave higher efficiency values than expected. Hence, it is advisable to further test this draft tube. For elbow type draft tube, efficiency of the system reached as high as 81.6 % but at a higher flow rate value i.e. 8.49 kg/s. In areas of variable discharge (part-load), elbow draft tube can be used because of lesser variations in efficiency at different flow rates.

10. References

1. Key World Energy Statistics. Date accessed: 20/03/2017. Crossref.
2. Gubin MF. Draft Tubes of Hydro Electric Stations. Amerind Publishing Company Co Pvt. Ltd. India, 1973.
3. Date accessed: 20/03/2017. Crossref.
4. Khare R, Prasad V, Mittal SK. Effect of Runner Solidity on Performance of Elbow Draft Tube, *Energy Procedia*. 2012; 14:2054–59. Crossref.
5. Khare R, Prasad V, Verma M. Design Optimisation of Conical Draft Tube of Hydraulic Turbine, *International Journal of Advances in Engineering Science and Technology*. 2012 May; 2(1):21–26.
6. Jeon JH, Byeon SS, Kim YJ. Effects of Draft Tube on The Hydraulic Performance of A Francis Turbine, *Materials Science and Engineering*. 2013; 52:1–7. Crossref.
7. Prasad V, Khare R, Chincholikar A. Hydraulic Performance of Elbow Draft Tube for Different Geometric Configurations using CFD. *IGHEM*, 2010, p. 252–56. PMID: 20064539.
8. Soni V, Roghelia A, Desai J, Chauhan V. 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, p. 1–10.
9. Duquesne P, Maciel Y, Dan Ciocan G, Deschênes C. Flow Separation in a Straight Draft Tube Particle Image Velocimetry. *IOP Conference Series Earth and Environmental Science*. 2014 Sep, p. 2–11.
10. Bhatt GB, Shah DB, Patel KM. Design Automation and CFD Analysis of Draft Tube for Hydro Power Plant, *International Journal of Mechanical and Production Engineering*. 2015; 3(6):40–43.
11. Anup KC, Lee YH, Thapa B. CFD Study on Prediction of Vortex Shedding in Draft Tube of Francis Turbine and Vortex Control Techniques, *Renewable Energy*. 2016; 86:1406–21. Crossref.
12. Jacob T. Evaluation Sur Modèle Réduit Et Prédiction De La Stabilité De Fonctionnement Des Turbines Francis. *École Polytechnique Fédérale De Lausanne*. 1993, p. 1–226.
13. Khurana S, Singh N, Singh H. Effect of Cavitations on Hydraulic Turbines- A Review, *International Journal of Current Engineering and Technology*. 2012; 172–77.
14. Thicke, RH. Practical Solutions for Draft Tube Instability, *Water Power and Dam Construction*. 1981; 33(2):31–37.
15. Brekke H. A Review of some Dynamic Problems in Hydropower Plants. *International Association for Hydro-Environment Engineering and Research WG1*, 2003.
16. Blommaert G. Étude du Comportement Dynamique des Turbines Francis Contrôleactif de Leur stabilité de Fonctionnement. *École Polytechnique Fédérale de Lausanne*. 2000, p. 1–151.
17. Resiga RS, Thi CVU, Muntean S, Ciocan GD, Nennemann B. Jet Control of the Draft Tube Vortex Rope in Francis Turbines at Partial Discharge. *23rd IAHR Symposium*. 2006 Oct, 1(14). PMID:17308691.
18. Chen C, Nicolet C, Yonezawa K, Farhat M, Avellan F, Tsujimoto Y. One-Dimensional Analysis of Full Load Draft Tube Surge, *Journal of Fluids Engineering*. 2008; 130. Crossref.
19. Miyagi O. Cavitations in and Consequent Vibration of the Draught Tube of a Water Turbine, *Journal of the Society of Mechanical Engineers Japan*. 1930; 33(2):57–59.
20. Brennen CE. *Bubble Dynamics Damage and Noise. Hydrodynamics of Pump*. Cambridge University Press. 2011, p. 78–95. Crossref.