IJESRT

INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

DESIGN AND EVALUATION OF PERFORMANCE OF CONICAL TYPE DRAFT TUBE WITH VARIATION IN LENGTH TO DIAMETER RATIO

Umashankar Nema*, Dr. Rohit Rajvaidya
** M.Tech Scholar, University Institute of Technology, Barkatullah University, Bhopal-462026
Assistant Professor, University Institute of Technology, Barkatullah University, Bhopal-462026*

ABSTRACT

Reaction turbines are widely used in medium and low head hydro power plants. Draft tube is one of the major components for reaction turbines. Water after passing through the runner has very high kinetic energy but lags in pressure energy. To recover pressure head and utilize kinetic energy of water, draft tubes are installed at outlet of turbine. Draft tube can be of gradually increasing cross-sectional type (commonly called conical diffuser type), simple elbow type or elbow with varying cross- section. Conical diffuser type draft tubes are simple in construction, simple to install, simple in designing and easy to maintain. The diffuser angle is less than Ten Degree to prevent flow separation and cavitation.

The length to inlet diameter ratio is one of the most important parameter in designing of draft tube. For variation in the ratio, inlet diameter of draft tube cannot be altered as variation in diameter may not fit site conditions. So the only length of draft tube can be varied for finding the best optimum length.

In present work, three conical diffuser type draft tube with length (22.45 m, 30.4 m and 38.4 m) have been modeled with a diffuser half angle of (4°) The velocity of water at inlet of draft tube has been kept as 9.17 m/s in axial direction, in radial direction as 0.14215 m/s and in tangential direction as 4.35 m/sec. The outlet of draft tube has been defined as outlet to atmosphere. The performance of all three draft tubes have been evaluated using ANSYS CFD. Velocity and pressure contours have been obtained for all the cases. It has been found that with the help of CFD, fluid flow problems can be easily handled. For present considered cases, the draft tube with length 30.4 m is best suited. The efficiency is found out to be 22.7% for both 30.4 m and 38.4 m draft tubes

INTRODUCTION

HYDRAULIC TURBINES

In hydraulic turbines series of blades are fitted to rotating shaft. Flowing water when pass through hydraulic turbine, strikes blades of the turbine and makes shaft to rotate. The velocity and pressure of water changes while flowing through turbines. This leads to the development of torque and rotation of shaft of turbine. There are various forms of hydraulic turbines in use depending on requirements. Particular type of turbine is used for specific need.

Classification of Hydraulic Turbines
Hydraulic turbines may be classified on the basis of:
- **Direction of flow of water**
- **Pressure change**
- **Head and quantity of water required**
- Position of the turbine shaft
- Specific speed
- Based on the direction of flow of water

Based on the flow path of the liquid hydraulic turbines can be categorized into three types.

   a. **Tangential flow hydraulic turbines: W**ater flows along the tangent to path of rotation of runner. Pelton turbine is a tangential flow turbine.
   b. **Axial flow hydraulic turbines:** The flow path of the liquid of these types of turbines is mainly parallel to the axis of rotation. Kaplan turbine has flow mainly in axial direction.
   c. **Radial flow hydraulic turbines:** These hydraulic turbines have the liquid flowing mainly in a plane perpendicular to the axis of rotation.
   d. **Mixed flow hydraulic turbines: T**urbines in which there is a flow component in both axial and radial direction are mixed flow turbines. Francis turbine is an example of mixed flow type. In Francis turbine water enters in radial direction and exits in axial direction.

Based on pressure change

One important criterion for classification of hydraulic turbines is change in pressure of liquid while it flowing through the rotor of the hydraulic turbines. Based on the pressure change hydraulic turbines can be classified as of two types.

   a. **Impulse turbine:** In an impulse turbine, all the available energy of water is converted into kinetic energy or velocity head by passing it through a contracting nozzle provided at the end of penstock. The pressure of liquid does not change while flowing through the rotor of the machine. Water is in contact with only a part of runner at a time and throughout its action on the runner and in its subsequent flow to tail race, the water is at atmospheric pressure. Example of impulse turbine is Pelton Wheel.
   b. **Reaction turbine:** The pressure of liquid changes while it flows through the rotor of the machine. The change in fluid velocity and reduction in its pressure causes a reaction on the turbine blades; this is where from the name reaction. Examples are – Francis, Kaplan turbines.

- *On the basis of head and quantity of water required*
a. High head turbines: Turbines which are capable of working under very high heads ranging from several hundred of metres are high head turbines. These turbines thus require relatively less quantity of water.
b. Medium head turbines: The turbines which are capable of working under heads ranging from 50 m to 250 m are medium head turbines. These turbines require relatively larger quantity of water.
c. Low head turbines: Turbines which are capable of working under the heads less than 50 m are low head turbines. These turbines require large quantity of water.
- *On the basis of position of the turbine shaft*
a. Vertical axis turbine – The axis of turbines is vertical.
b. Horizontal axis turbine – The axis of turbines is horizontal.

On the basis of Specific speed

   a. **Low specific speed turbines:** Turbines whose specific speed varies between 8 to 35 are low specific speed turbines.
   b. **Medium specific speed turbines:** Turbines working under the range 35 to 400 are medium specific speed turbines.
   c. **High specific speed turbines:** Turbines whose specific speed varies from 400 to 1000 are high specific speed turbines.

The specific speed of a turbine is defined as the speed of a geometrically similar turbine that would develop 1 kW under 1 m head. All geometrically similar turbines (irrespective of the sizes) will have the same specific speeds when operating under the same head.

$$Specific\ Speed, n_s = \frac{n\sqrt{P}}{H^{5/4}}$$

Reaction Turbines

At entrance to the runner only a part of available energy is converted into kinetic energy and remaining part remains in the form of pressure energy. As water flows through runner, change from pressure energy to kinetic energy takes place. Pressure energy at inlet to the turbine is much higher than pressure at outlet. Variation in pressure throughout the passage of water is there. For this gradual change, runner is completely enclosed in an air tight casing and the casing is fully filled with water throughout operation of turbine. The difference in pressure at inlet and outlet of turbine is called reaction pressure and hence these turbines are called reaction turbines.

Water from penstock enters spiral casing which is around the runner. Casing is provided for evenly distribution of water at constant velocity. Too keep velocity of water constant, area of casing is gradually decreased. Casing is made up of cast steel, plate steel, concrete depending upon the pressure to which it is subjected. After spiral casing water passes to stay rings. Stay ring consists of stay vanes held together by upper and lower ring. Stay ring directs water from casing to guide vanes and it also resists load imposed upon it by internal pressure of water and weight of turbine. Stay rings are made up of cast iron or cast steel. After stay rings water passes through guide vanes which are provided around turbine runner. Guide vanes regulate the quantity of water passing through them at appropriate angle. Guide vanes are aerofoil shaped and are made up of cast steel of stainless steel. Main purpose of all the above described components is to deliver water to runner with least loss of energy.

Water then passes through runner where transformation of energy takes place. Torque produced by the runner is transmitted to generator through the shaft which is connected to generator shaft. Water after passing through runner passes to tail race through draft tube. Draft tube is a pipe of gradually increasing cross-sectional area which connects runner to exit of tail race. It is made of concrete or cast steel. It is airtight and the lower end of it is submerged below the level of water in tail race. It has two functions to perform:
1. It permits suction head to be established at runner exit which makes turbine to be installed above tail race without loss of head.
2. It converts large portion of kinetic energy rejected from runner into useful pressure energy.

Fig.1.1: Components of reaction turbine

Commonly used draft tubes are:
1. Straight divergent type draft tube
2. Moody spreading type
3. Simple elbow draft tube
4. Elbow type draft tube having circular cross section at inlet and rectangular cross   section at outlet.
Choosing the type of draft tube depends upon available space for installation.

LITERATURE REVIEW

Computational Fluid Dynamics (CFD) emerged with continuous development in speed of computers and memory size with the development of accurate numerical methods CFD is a computer-based tool for simulating the behavior of systems involving fluid flow, heat transfer, and other physical processes.

A lot of work and studies have been done on turbines and its hydraulic components using CFD which are described in various research papers. Few of the papers which are referred are described below.

Brekke H., "Performance and safety of hydraulic turbines" [1], has discussed about turbine selection, optimisation of performance of turbines and its operation and maintenance with special attention to bolt connections.

Vishnu Prasad, et. al., "Numerical Simulation for Performance of Elbow Draft Tube at Different Geometric Configurations" [2] have numerically calculated efficiency and losses from pressure and velocity distributions for elbow draft tube.

Vishnu Prasad, et. al., "Hydraulic Performance of Elbow Draft Tube for Different Geometric Configurations using CFD" [3] have varied length and height at different mass flow rates of elbow draft tube. Results are also compared with experimental values.

Monica Sanda LLiescu, et. al., "Analysis of the Cavitation Draft Tube Vortex in a Francis Turbine Using Particle Image Velocimetry Measursments in Two- Phase Flow" [4] have used particle image velocimetry system to predict cavitation in Francis turbine.

V De Henau et.al., "Computational Study of a Low Head Draft Tube and Validation with Experimental Data" [5] have investigated reliability of CFD analysis of low head turbine draft tubes. CFD results are compared with experimental results. They concluded that specifying only average values over inlet region can be sufficient for finding efficiency of draft tube.

Yodchai Tiaple et.al., "*The Development of Bulb Turbine for Low Head Storage Using CFD Simulation*"[6], FLUENT software is selected for determining flow pattern. Hydro Turbine for Lower Mae Ping dam has been designed considering existing civil structure of the dam, flow regulation for irrigation and the limitation of water level that can effect to the efficiency of hydro power plant at upper dam. The optimizations for all purposes have been considered.

J H Jeon et.al., *"Effect of Draft Tube on the Hydarulic Performance of a Francis Turbine"* [7] have numerically investigated performance of Francis turbine with three different types of draft tubes and thickness of guide vanes. Eisinger et.an., in "*Automatic shape optimization of hydro turbine components based on CFD*"[8], have designed draft tube of various cross-section. Different optimization algorithms have been applied and discussed.

Kearon Bennett et.al., in "*Application of CFD Turbine Design for Small Hydro Elliott Falls, A Case Study*"[9], have describes the original design of the power plant, the problems encountered, designing of new stay vanes, hub and blades, the solutions adopted, and the results of the repair/upgrade using CFD software.

Ian Padureaneles et.al., in "" [10] have presented some calculation related to the hydraulic losses in the spiral casing, stay ring and draft tube of the hydraulic Francis turbine. Numerical results are compared to values of hydraulic losses found in literature and observed similarity.

In all the above papers CFD software has been used to optimize the design of the turbines and their components to get the desired result. In this dissertation CFD software has been used to check the performance of three lengths of draft tube at same mass flow rates of water.

METHODOLOGY
COMPUTED PARAMETERS
Head loss:

$$H_{LD} = \frac{TP_{in} - TP_{out}}{\rho g}$$

2.   Head loss coefficient:

$$\xi = \frac{\rho g H_{LD}}{TP_{in}}$$

3.   Head recovery:

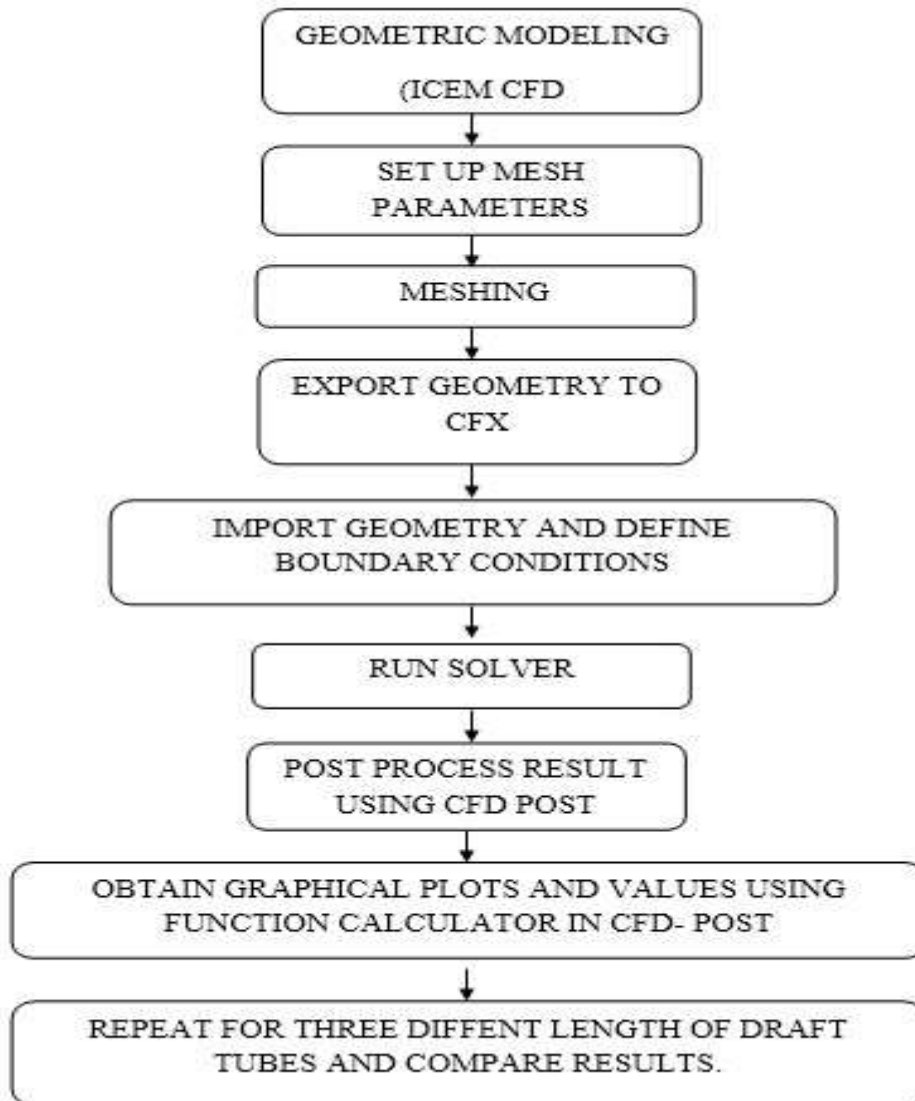$$\Delta h_d = \left(\frac{v_1^2}{2g} - \frac{v_2^2}{2g} \right) - H_{LD}$$

4.   Head recovery coefficient:

$$\zeta = \frac{\left\{ \left(\frac{\rho v_1^2}{2} - \frac{\rho v_2^2}{2} \right) - \rho g H_{LD} \right\}}{TP_{in}}$$

5.   Efficiency:

$$\eta_D = \frac{\Delta h_d}{\frac{v_1^2}{2g}} X 100$$

FLOW CHART FOLLOWED FOR SIMULATION

```
        ┌─────────────────────────┐
        │   GEOMETRIC MODELING    │
        │      (ICEM CFD          │
        └───────────┬─────────────┘
                    │
        ┌───────────▼─────────────┐
        │     SET UP MESH         │
        │      PARAMETERS         │
        └───────────┬─────────────┘
                    │
        ┌───────────▼─────────────┐
        │        MESHING          │
        └───────────┬─────────────┘
                    │
        ┌───────────▼─────────────┐
        │  EXPORT GEOMETRY TO     │
        │         CFX             │
        └───────────┬─────────────┘
                    │
    ┌───────────────▼───────────────┐
    │ IMPORT GEOMETRY AND DEFINE    │
    │    BOUNDARY CONDITIONS        │
    └───────────────┬───────────────┘
                    │
        ┌───────────▼─────────────┐
        │      RUN SOLVER         │
        └───────────┬─────────────┘
                    │
        ┌───────────▼─────────────┐
        │  POST PROCESS RESULT    │
        │    USING CFD POST       │
        └───────────┬─────────────┘
                    │
  ┌─────────────────▼─────────────────┐
  │ OBTAIN GRAPHICAL PLOTS AND VALUES │
  │   USING FUNCTION CALCULATOR IN    │
  │            CFD- POST              │
  └─────────────────┬─────────────────┘
                    │
  ┌─────────────────▼─────────────────┐
  │ REPEAT FOR THREE DIFFENT LENGTH   │
  │ OF DRAFT TUBES AND COMPARE        │
  │           RESULTS.                │
  └───────────────────────────────────┘
```

RESULTS AND DISCUSSION
GENERAL
Three different length of draft tube with length 22.45 m, 30.4 m and 38.4 m (length to diameter at inlet with ratio 14, 19, 14) have been analysed and their performance has been evaluated at constant mass flow rates. Velocity component in axial direction = 9.17 m/s, radial component= 0.142151 m/s and tangential component= 4.35075 m/sec (Ruchi 2011) have been taken into consideration.

Each draft tube has been taken a domain of fluid type and fluid is taken as water with reference pressure as 0 atmospheric. Heat transfer option has been set to none, fluid temperature 25°C and turbulence model SST κ-ω has been taken for the domain. Density of water has been defined as 997 kg/m^3 and kinematic viscosity as 0.8926 X $10^{-6} m^2/s$. The outlet of draft tube reference pressure is set equal to 1 atmospheric. Surface of draft tube is taken to be smooth and no slip condition is taken.

High resolution advection scheme with high resolution turbulence numerics for 500 iterations were given. The timescale control was set to Auto Timescale. The RMS residual target was set to 1x10E-8 for termination of the calculations. After completion of the iterations results are obtained. The variation of the pressure and velocity using pressure contours and velocity streamlines respectively on the surface of the draft tube could be observed.

COMPUTED PARAMETERS

The average values of velocity, pressure, total pressure at inlet and outlet of draft tube were obtained using the function calculator in CFX Post. Then efficiency of draft tube is calculated. The computed results in the tabular form have been shown in Tables 4.1 to 4.3 for the considered domain.

RESULTS

Draft tube with length 22.45 m (length/ diameter ratio 14)

Pressure In Pascal	Velocity In meter per second	Total pressure In Pascal	Head loss Meter	Head loss coefficient	Head recovery Meter	Efficiency
57784.7	10.101	108651	0.507	0.046	4.48	22.39%
101479	2.03755	103674				

Draft tube with length 30.4 m (length/ diameter ratio 19)

Pressure In Pascal	Velocity In meter per second	Total pressure In Pascal	Head loss Meter	Head loss coefficient	Head recovery Meter	Efficiency
57136	10.093	107923	0.565	0.051	4.532	22.70%
101404	1.362	102377				

Draft tube with length 38.4 m (length/ diameter ratio 24)

Pressure In Pascal	Velocity In meter per second	Total Pressure In Pascal	Head loss In meter	Head loss coefficient	Head recovery Meter	Efficiency
57058	10.099	107903	0.613	0.056	4.534	22.7%
101364	1.002	101892				

For all the cases, pressure at inlet is low and velocity is high whereas at outlet, pressure is high and velocity low which shows conversion of velocity head into pressure head. Total pressure is higher at inlet as compared to outlet for all the cases.

As the length of draft tube increases, head loss and head loss coefficient increases. Head recovery and efficiency increases. But for draft tube with length 30.4 m and 38.4 m, efficiency comes out to be nearly constant.

Installing draft tube of longer length will lead to increase in material cost. So draft tube with length 30.4 m is appropriate for considered case.

GRAPHICAL PLOTS

Pressure contours and velocity stream lines were obtained using insert contour and insert streamline commands of menu bar in ANSYS CFX-Post. Pressure contours and velocity stream lines for each geometry have been shown under.
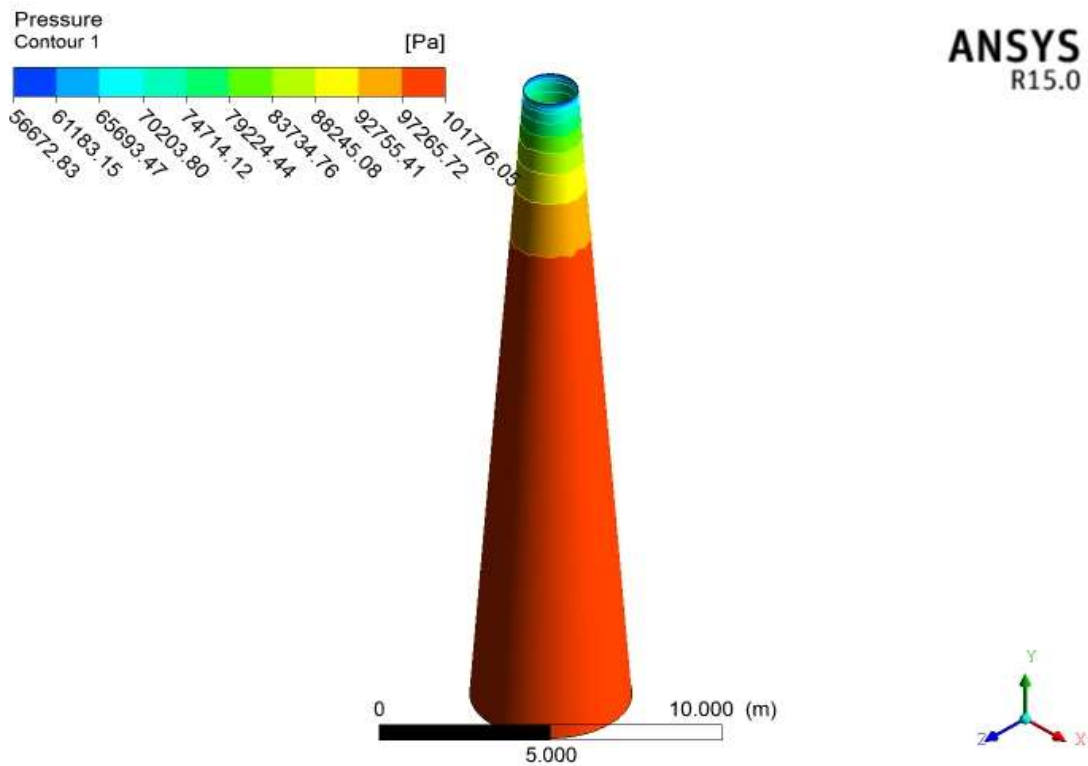
For 1st Draft tube

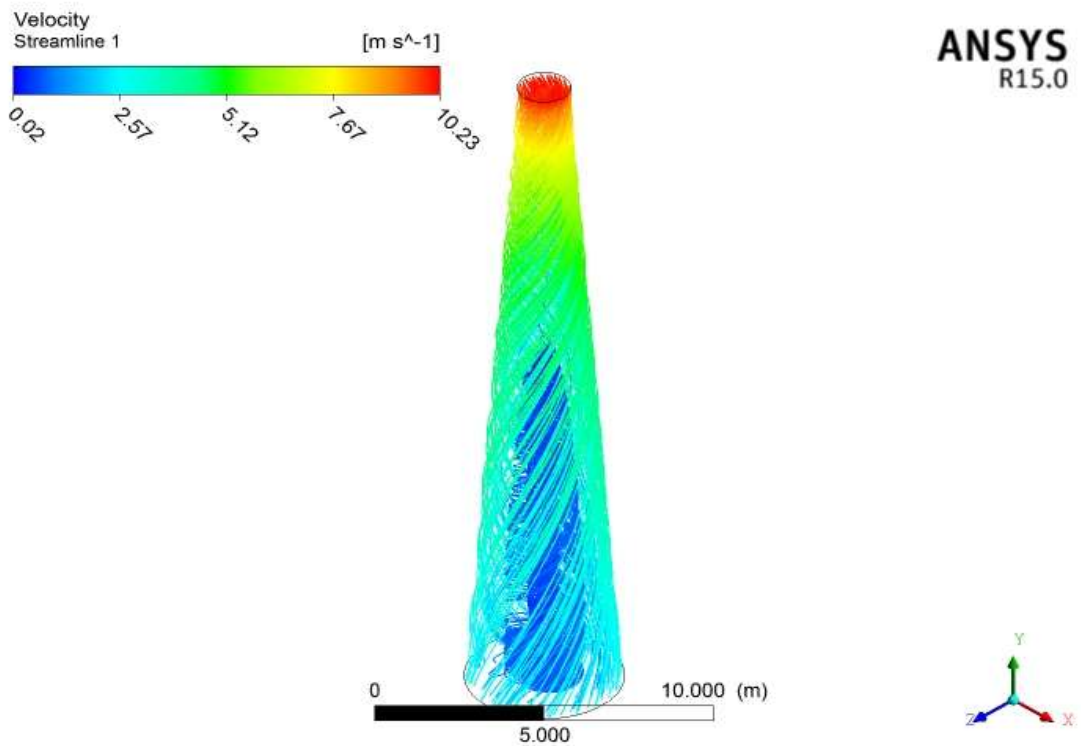

Fig 4.1: Pressure contour at 1st draft tube

Fig 4.2: Streamlines showing the velocity distribution in 1st geometry
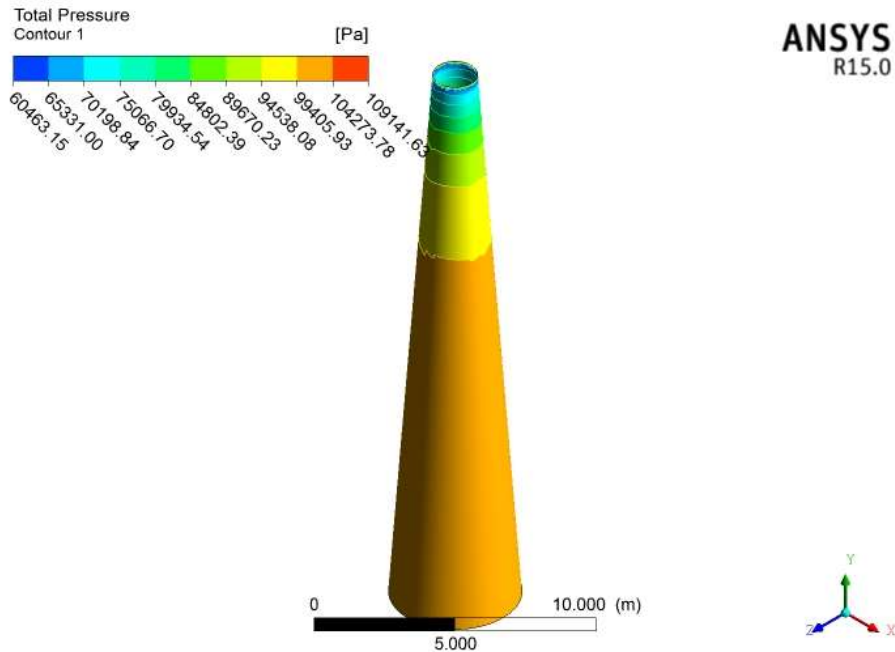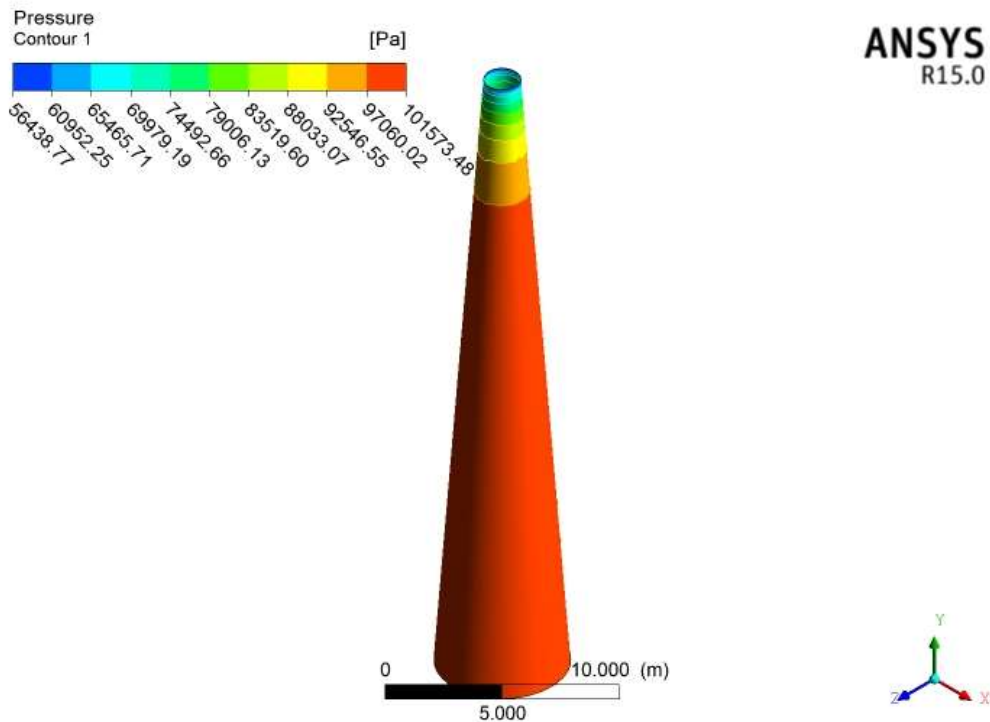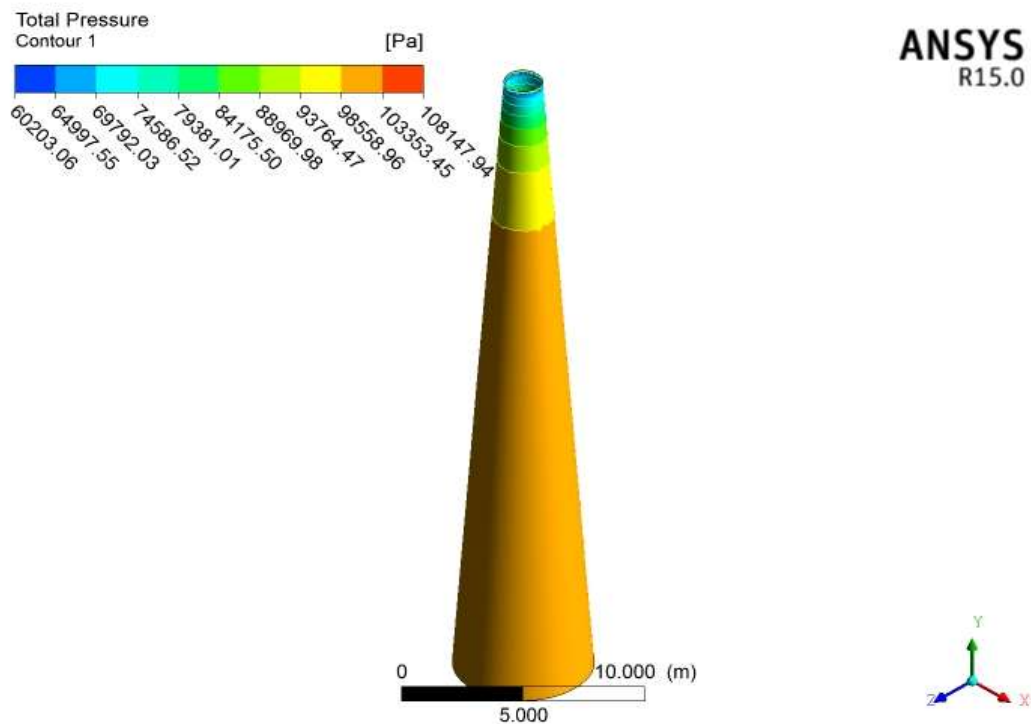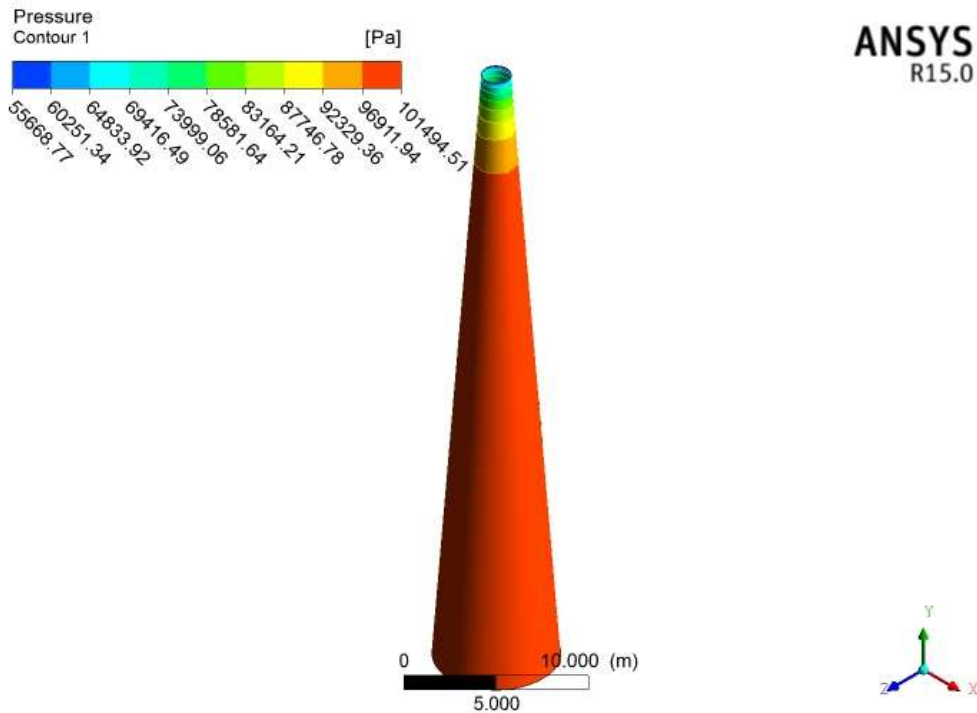
Fig 4.3: Variation in total pressure in 1ˢᵗ geometry

From Fig. 4.1, it is observed that pressure from inlet to outlet of draft tube increases. Velocity decreases from inlet to outlet as seen in Fig. 4.2. Lowest velocity is observed in mid of exit of draft tube. From Fig. 4.3, it is seen that there is much variation in total pressure at inlet of draft tube as compared to outlet.

For 2ⁿᵈ Draft tube

Fig 4.4: Pressure contour at 2ⁿᵈ draft tube

THOMSON REUTERS
ENDNOTE

[Nema* *et al.,* 6(2): February, 2017]
IC™ Value: 3.00

ISSN: 2277-9655
Impact Factor: 4.116
CODEN: IJESS7
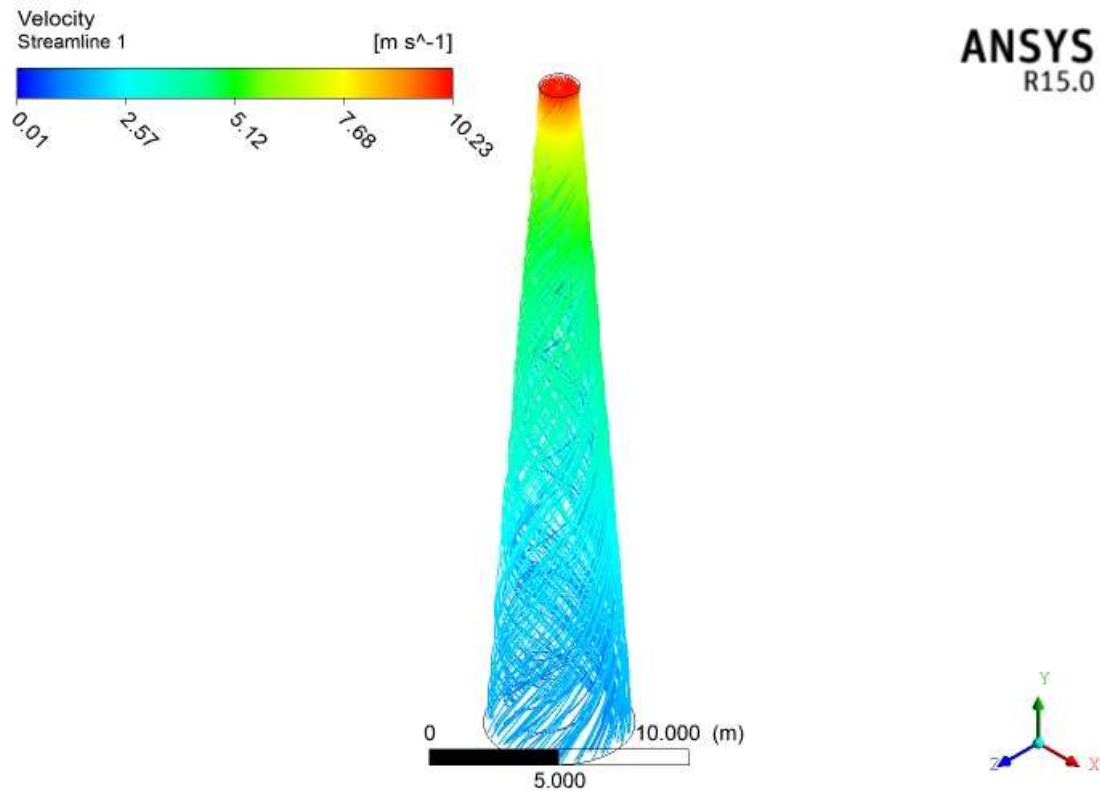
Fig 4.5: Streamlines showing the velocity distribution in 2nd geometry,

Fig 4.6: Variation in total pressure for 2nd geometry

From Fig. 4.4 and Fig. 4.6, variation in pressure and total pressure at surface of draft tube can be observed. Water velocity streamlines can be seen in Fig. 4.5. Velocity of water decreases from inlet to exit of draft tube.

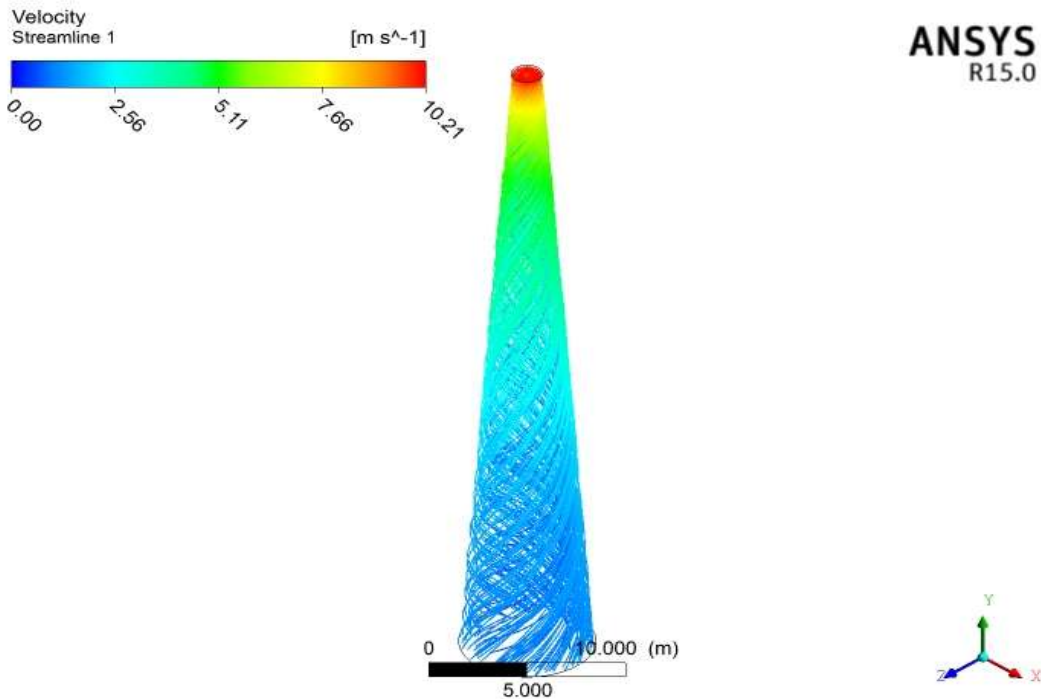**For 3rd Draft tube**

Fig 4.7: Pressure contour at 3rd domain

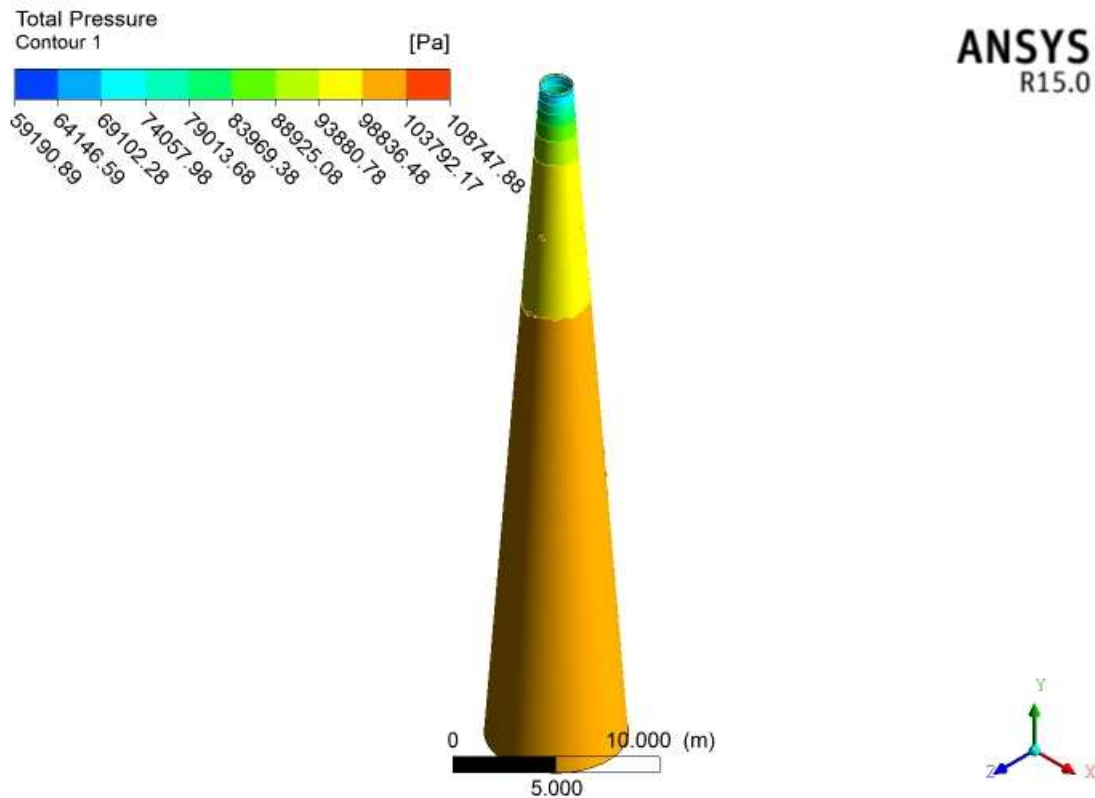*Fig 4.8: Streamlines showing the velocity distribution in 3rd geometry*

Fig 4.9: Variation in total pressure for 3rd geometry

CONCLUSION & FUTURE SCOPE

The computation and comparison of different flow coefficients of various geometric configurations using CFD will help to optimize the draft tube. In this dissertation Computational Fluid Dynamics (CFD) approach has been used to predict the performance of different length of draft tubes at same velocity components.

Following conclusion has been made from above work:

1. The pressure at the inlet of the draft tube is less than pressure at outlet. This confirms that pressure is regained in draft tube.
2. The pressure is much lower at the inlet than at the outlet of the draft tube which is equal to atmospheric pressure. The outlet velocity of the water is lower than the inlet velocity of the fluid which shows that the kinetic energy is being converted into pressure energy.
3. With the increase in length of draft tube, the variation in velocity increases along with variation in total pressure. The exit velocity decreases for the same mass flow rate. The head loss and head loss coefficient is found out to be highest for longest draft tube.
4. Similar kind of water velocity streamlines are found out to be for all the cases. Swirl motion of water is seen in all the cases.
5. From the streamlines, it is seen that the water at mid of the draft tube has least velocity For longest draft tube, the velocity of water at mid of exit is nearly zero.
6. In present case it may be concluded that the draft tube with length 30.4 m with length to diameter ratio 24 is better than the other two draft tubes.

FUTURE SCOPE OF WORK

- The shape of draft tube and its effect on the losses, streamlines and pressure contours can be analysed.
- The work can be extended for various draft tube angles also..
- The work can be extended for analyzing other parts of reaction turbines also.

REFERENCE

[1] Brekke H., "Performance and safety of hydraulic turbines" [1], has discussed about turbine selection, optimisation of performance of turbines and its operation and maintenance with special attention to bolt connections.

[2] Prasad V., Khare R., Chincholikar, 2011, Numerical simulation for performance of elbow draft tube at different geometric configurations, AHEC, IIT Roorkee

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

[4] Iliescu Monica Sanda, Ciocan Gabriel Dan, Avellan François, 2008, Analysis of the Cavitating Draft Tube Vortex in a Francis Turbine Using Particle Image Velocimetry Measurements in Two-Phase Flow, Journal of Fluids Engineering, Vol. 130, DOI: 10.1115/1.2813052.

[5] Henau V De, Payette F A, Sabourin M, Deschênes C, Gagnon J M, Gouin P, 2010 Computational study of a low head draft tube and validation with experimental data, 25th IAHR Symposium on Hydraulic Machinery and Systems, doi:10.1088/1755-1315/12/1/012084

[6] Choi Hyen-Jun, Zullah Mohammed Asid, Roh Hyoung-Woon, Ha Pil-Su, Oh Sueg-Young, 2013, "CFD Validation of Performance of a 500 kW Francis Turbine", Renewable Energy 54, pp 111-123.

[7] Motycak Lukas, Skotak Ales, Obrovsky Jiri, 2010, "Conditions of Kaplan Turbine CFD Analysis", ANSYS Conference.

[8] M.F Gubin, 1973, "Draft Tubes of Hydraulic Station", Amerind Publishing Company Pvt.Ltd.

[9] Khare Ruchi, Prasad Vishnu, Mittal Sushil Kumar, 2012, "Effect of Runner Solidity on Performance of Elbow Draft Tube", Energy Procedia 14, pp 2054-2059.

[10] Khare Ruchi, Prasad Vishnu, Kumar Sushil, 2010, "Derivation of Global Parametric Performance of Mixed Flow Hydraulic Turbine Using CFD", Hydro Nepal.

[11] U. Andersson, J. Jungstedt and M.J. Cervantes, 2008, "Model Experiments of Dynamic Loads on a Draft Tube Pier", 24th Symposium on Hydraulic Machinery and Systems.

[12] S Tridon, S Barre, G D Ciocan, P Leroy and C Segoufin, 2010, "Experimental investigation of draft tube flow instability", 25th IAHR Symposium on Hydraulic Machinery and Systems.

[13] ANSYS, 2015, "ANSYS CFX 15 Software Manual, ANSYS, Inc., southpointe, Canonsburg, PA.

[14] P. K. Sinha, A.K.das and B. Majumdar,, "Numerical Investigation of Flow Through Annular Curved Diffuser", International Journal of Mechanical Engineering & Technology (IJMET), Volume 2, Issue 2, 2011, pp. 1 - 13, ISSN Print: 0976 – 6340, ISSN Online: 0976 – 6359.

[15] Prof. N.V.Hargude and Prof. P.R.Patil, "Design and Development of Plc Operated Hydraulic System for Machining Process", International Journal of Mechanical Engineering & Technology (IJMET), Volume 3, Issue 2, 2012, pp. 366 - 373, ISSN Print: 0976 – 6340, ISSN Online: 0976 – 6359.

[16] Tarun Singh Tanwar , Dharmendra Hariyani and Manish Dadhich, "Flow Simulation (CFD) & Static Structural Analysis (FEA) of a Radial Turbine", International Journal of Mechanical Engineering & Technology (IJMET), Volume 3, Issue 3, 2012, pp. 252 - 269, ISSN Print: 0976 – 6340, ISSN Online: 0976 – 6359. 1. Brekke H., 2010, Performance and safety of hydraulic turbines, 2010 IOP Conf. Ser.: Earth Environ. Sci. 12 012061