International Journal of Science and Research (IJSR)
ISSN (Online): 2319-7064
Index Copernicus Value (2016): 79.57 | Impact Factor (2017): 7.296

Design and CFD Analysis of Conical Shape Draft Tube

Vikas Rai[1], Nishant Vibhav Saxena[2]

[1]M.Tech Scholar, Department of Mechanical Engineering,
Millennium Institute of Technology, Bhopal, M.P., India
[1]raivikas51[at]gmail.com

[2]A.P. and Head, Department of Mechanical Engineering,
Millennium Institute of Technology, Bhopal, M.P., India
[2]nishant.mgi[at]gmail.com

Abstract: ***For power generation, in medium head hydro power plants, reaction turbines are used. Water leaving the reaction turbine has high kinetic energy and low pressure energy. To recover the energy, draft tubes at exit of runners are used. Conical draft tubes are the simplest form of draft tube and it has high efficiency. In the present work, geometric modeling of a conical shape of draft tube has been done. Meshing has been done in ICEM CFD. Various parameters have been found out for specified boundary conditions. From results it has been found that as the water moves out of the draft tube, kinetic energy of water is reduced and pressure energy is recovered.***

Keywords: Reaction turbines, Head Loss, Efficiency, Draft Tube, Head Recovered, Pressure, Velocity.

1. Introduction

Reaction turbines have a series of blades fitted to the rotating disk. Flowing water, after passing through the runner has high kinetic energy. But it possesses very low pressure energy. The kinetic energy can be used and pressure energy can be recovered using draft tube. Conical draft tube is one of the most simple of its kind. The conical draft tube is a pipe of gradually increasing cross sectional area. Gradual increase in cross-section should be such that the divergent angle should not be more than 5°. In the present work, CFD analysis of a draft tube has been done for existing boundary conditions specified by Ruchi et al. [1].

2. Literature Review

Brekke H., [2] 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., [3] have numerically calculated efficiency and losses from pressure and velocity distributions for elbow draft tube. Ruchi Khare, et. al., [4] have varied length and height at different mass flow rates of elbow draft tube. Results are also compared with experimental values. In above papers, CFD software ANSYS has been used to optimize the design of the turbines and their components to get the results.

3. Geometric Modeling and Meshing

3.1 Geometric Modeling

Geometry for draft tube has been modeled. The flow domain consists of inlet, draft tube surface, outlet and flow domain. Flow domain contains water. The draft tube surface of the domain is specified as wall.

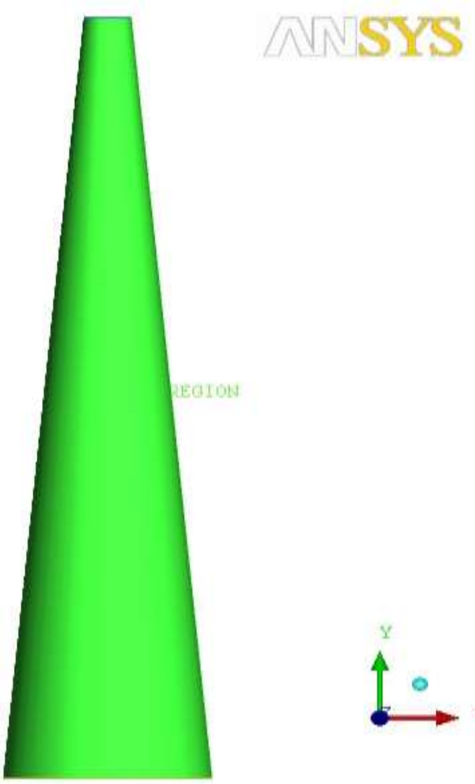

Figure 1: Isometric view of Draft tube

3.2 Meshing

Draft tube domain is enclosed by inlet and outlet surfaces. The velocity components specified by Ruchi et.al. (2012) have been specified at inlet to draft tube. The outlet of draft tube is kept open to atmosphere.

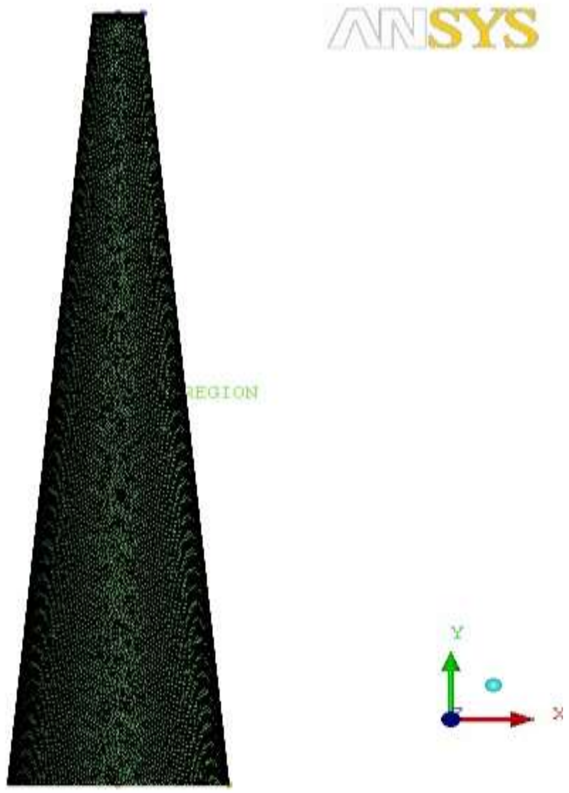

Figure 2: Mesh of Flow Domain

Table 1: No. of Mesh Elements

Part name	No. of Elements	Element Type
Inlet	5194	Triangle
Outlet	6275	Triangle
Draft Tube Wall	77043	Triangle
Flow Region	3426741	Tetrahedral

3.3 Boundary Conditions

- Inlet Boundary Condition: The water velocity in axial, tangential and radial direction is specified at inlet of draft tube. Turbulence is set to medium intensity with value of 5%.
- Outlet Boundary Condition: The pressure at the outlet of draft tube domain is set equal to 1 atmospheric.
- Wall Conditions: The surface of draft tube is assumed to be smooth wall with no slip condition.
- Turbulence Model: SST turbulence model with automatic wall function is applied as SST model can capture boundary layer better.

4. Results and Discussions

Draft tube with length 30.5 m have been analysed and its performance has been evaluated for fixed water velocity components. Velocity component in axial direction = 9.17 m/s, radial component= 0.142151 m/s and tangential component= 4.35075 m/sec (Ruchi et al., 2012) have been taken into consideration.

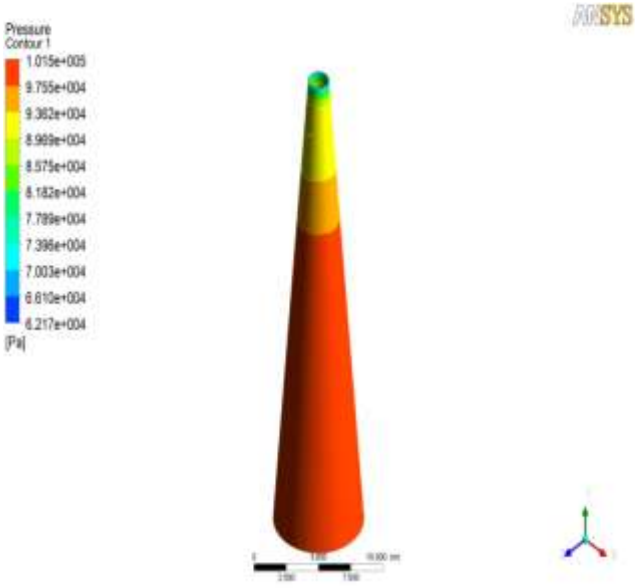

Figure 3: Pressure contour

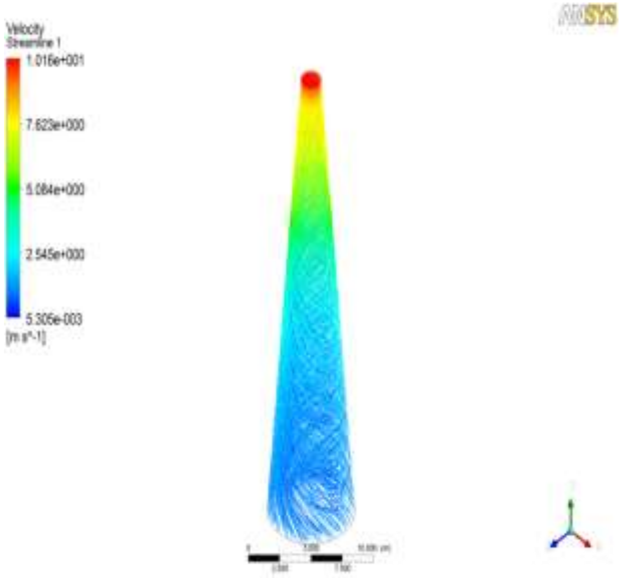

Figure 4: Streamlines showing the velocity distribution

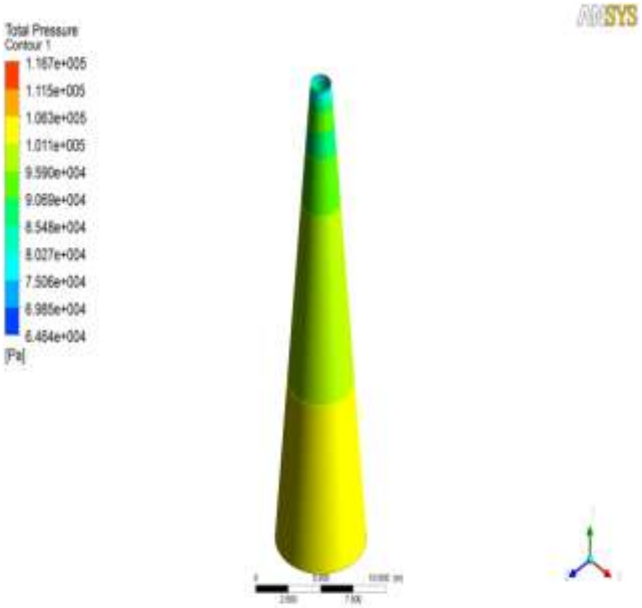

Figure 5: Variation in total pressure

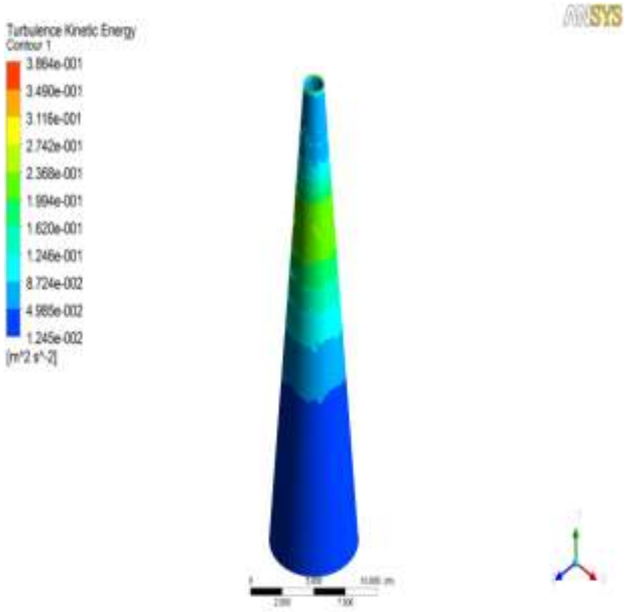

Figure 6: Variation in turbulent kinetic energy

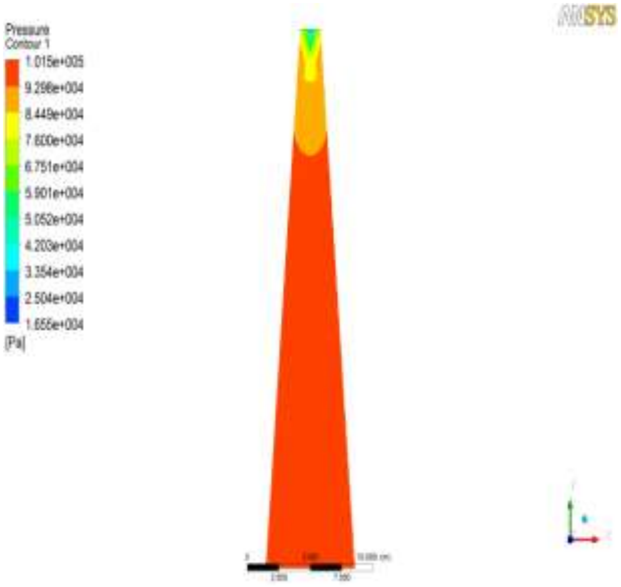

Figure 7: Pressure contour at mid section of draft tube

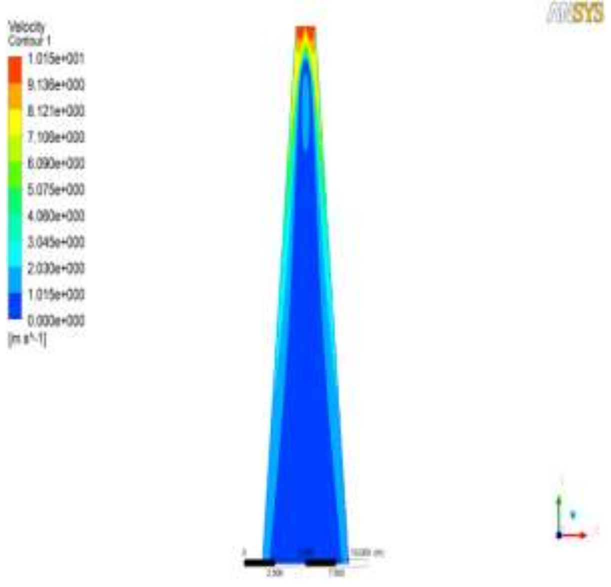

Figure 8: Velocity contour at mid section of draft tube

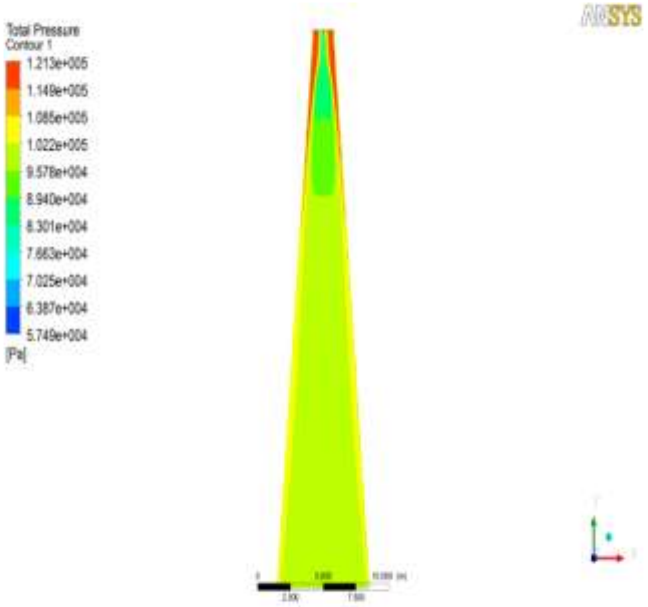

Figure 9: Total Pressure contour at mid section of draft tube

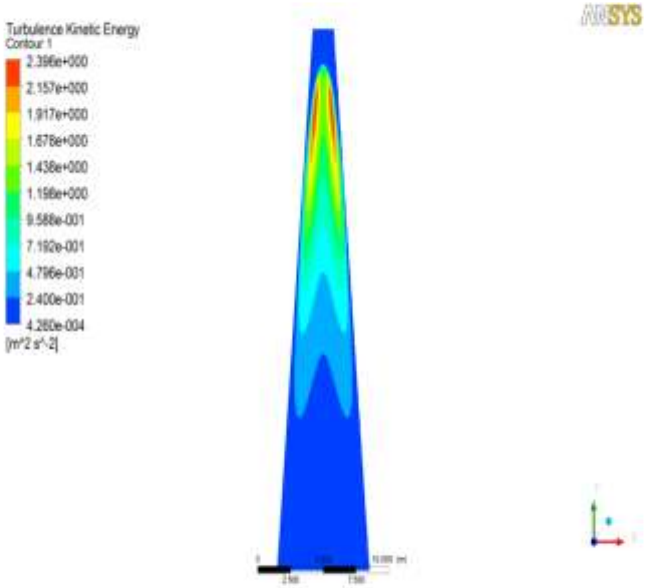

Figure 10: Turbulent kinetic energy contour at mid section of draft tube

Table 2: Values of Computed Parameters

S. No.	Parameters	Values
1	Head Loss	0.66
2	Head Loss Coefficient	0.06
3	Head Recovery	4.55
4	Efficiency	86.86

5. Conclusion

Pressure from inlet to outlet of draft tube increases. Velocity decreases from inlet to outlet of the draft tube. Lowest velocity is observed in mid section of exit of draft tube.

There is much variation in total pressure at inlet of draft tube as compared to outlet region. Value of turbulent kinetic energy is highest at the centre of the draft tube because of increase in cross-section of the draft tube. Highest turbulent kinetic energy is observed at mid volume of the draft tube.

Ratio of head loss to the inlet total pressure head is called head loss coefficient. It is non dimensional parameter. Head loss coefficient is 0.06 for the modeled geometry. The efficiency of the draft tube is found out to be 86.86%

References

[1] Khare, Ruchi, Vishnu Prasad, and Mitrasen Verma. "Design Optimisation of conical draft tube of hydraulic turbine." IJAEST International Journal of Advances in Engineering, Science and Technology 2.1 (2012): 22-26.
[2] Brekke, H. "Performance and safety of hydraulic turbines." IOP Conference Series: Earth and Environmental Science. Vol. 12. No. 1. IOP Publishing, 2010.
[3] Prasad, Vishnu, Ruchi Khare, and Abhas Chincholikar. "Hydraulic performance of elbow draft tube for different geometric configurations using CFD." IGHEM-2010, Ahec, IIT, Roorkee, India, October (2010).
[4] Khare, Ruchi, Vishnu Prasad, and Mitrasen Verma. "Design Optimisation of conical draft tube of hydraulic turbine." IJAEST International Journal of Advances in Engineering, Science and Technology 2.1 (2012): 22-26.
[5] ANSYS, 2012, "ANSYS CFX 12 Software Manual, ANSYS, Inc., southpointe, Canonsburg, PA.
[6] M.F Gubin, 1973, “Draft Tubes of Hydraulic Station”, Amerind Publishing Company Pvt. Ltd.
[7] Modi P.N., Seth S.M. , “Hydraulics and Fluid Mechanics”, Standard Book House, New Delhi.1991