

A Review on Advances in the Design and Analysis of Draft tube for Reaction turbines

Vikas Rai¹, Nishant Vibhav Saxena²

¹M.Tech Scholar, Department of Mechanical Engineering, Millennium Institute of Technology, Bhopal, M.P.

²A.P. and Head, Department of Mechanical Engineering, Millennium Institute of Technology, Bhopal, M.P.

Abstract - Draft tubes are one of the important component of hydroelectric power plants. Draft tube are conduits of gradually increasing cross-sectional area. Pressure head is recovered as water flows down the draft tube. In the present paper, work done by researchers has been discussed. It is observed that CFD can be used to predict the performance not of Draft tube rather whole hydraulic turbine components in design phase only. By using CFD, resources, time cost etc are saved.

Key Words: Reaction turbines, CFD, Draft Tube, Head loss

1. INTRODUCTION

Draft tube are the conduits of increasing cross sectional area which connect runner to the tail race. There are various types of draft tubes. Simplest one being conical type. The type of draft tube to be installed depends upon the space available for its installation. Because performance of draft tube directly affects the overall efficiency of hydro electric power plant, a lot of research has been done to optimise the design of the draft tube.

2. Literature Review

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 are described below:

T. Kubota and Yamada, S, 1982 have discussed the effect of cone angle on the characteristic of reaction turbine.

T. Kubota, Han F., Avellan F., 1996 have presented and applied the one-dimensional flow theory, a new algorithm of extracting the various component losses in bulb turbines from the performance diagrams measured with the model tests.

Gabriel Dan Ciocan et al. 2007 has studied experimentally and numerically the dynamics of the rotating vortex taking place in the discharge ring of a Francis turbine for partial flow rate operating conditions Unsteady RANS simulation have been performed and results have been compared to the experimental values. Both the numerical and experimental results bears a close comparisons.

Iliescu, M. S., et al. 2008 used particle image velocimetry (PIV) system for investigating the flow velocity field in the

case of a developing cavitation vortex at the outlet of a Francis turbine runner. The influence of the turbine setting level on the volume of the cavity rope has been investigated which provide a physical knowledge about the hydrodynamic complex phenomena involved in the development of the cavitation rope.

H. Altzibar, et al., 2008 have studied the performance of a draft tube conical spouted bed for drying fine particles in batch operations with nonporous, porous, and open-sided draft tubes in order to ascertain the optimum configuration of this internal device.

P. Gouin et al., 2009 have presented time averaged velocity profiles for both inlet and outlet sections of the draft tube and correlations are established between positioning system and rotating optical access design.

Prasad, V.et al. 2009 carried out 3D viscous turbulent flow analysis of an elbow draft tube. Length and height of the draft tube were varied along with mass flow rates. ANSYS CFX code was used. Efficiencies and losses are computed for the draft tube from pressure and velocity distributions and presented graphically. Obtained geometry for best performance bears close comparison to the geometry used in most of the hydro power stations.

Vishnu Prasad et al., 2010 computed draft tube efficiencies and losses from pressure and velocity distributions and presented graphically to study the effect of geometrical parameters on draft tube performance Ruchi Khare et al., 2010 for Francis turbine, the average values of flow parameters like velocities and flow angles at the inlet and outlet of runner, guide vane and stay vane of turbine are computed to derive flow characteristics.

Brekke, H. 2010 has discussed the choice of small turbines for run of the river type power plants. Optimization of the performance of different types of large turbines has also been discussed along with safety and necessary maintenance of turbines with special attention to bolt connections.

Romeo Susan-Resiga et al. 2010 discussed that the diffuser performance quickly deteriorates as the turbine discharge decreases, due to the occurrence and development of vortex breakdown. A novel flow control technique, which uses a water jet injected from the runner crown tip along the axis has been investigated.

Vishal Soni, et al. 2010 made various designs of bend type curved draft tube using conventional approach and CFD simulations are carried out at Best Efficiency Point and at off duty points of Francis turbine. From these different trials, an optimum draft tube is selected.

J.Makibar et al. 2011 studied the influence of temperature on conical spouted bed hydrodynamics and wall-to-bed and bed-to-surface heat transfer coefficients have been determined.

Ruchi Khare et al. 2012 carried out 3D viscous flow simulation in the complete flow passage of Francis turbine for three different runner solidities at different rotational speeds. The draft tube performance parameters have been computed from simulation results and effect of runner solidity and rotational speed on the performance of draft tube has been discussed. Numerical results are also compared with experimental results which bears a close agreement. Ruchi Khare et al. 2012 has specified that conical draft tubes gives better efficiency in comparison to elbow draft tubes because of more recovery of vortex flow coming out of runner. Numerical simulation of conical draft tube has been done using ANSYS CFX 13.0 for variation in length and diffuser angle of the draft tube. The performance of draft tube has been analyzed by calculating head loss, head recovery coefficients and efficiency of draft tube from simulation results.

Rahul Bajaj et al. 2014 found that the geometry of draft tube affects the performance of reaction turbine to large extent. Using computational power, alternative design of draft tube has been investigated. Numerical simulation has been done for a large elbow draft tube with pier in diffuser. The length of diffuser has been changed to see its effect on the performance of draft tube.

McNabb, J., et al. 2014 described an approach for dealing with this optimization problem. The draft tube's detailed geometry is defined as a function of a comprehensive set of design parameters. Analysis has been done using Navier-Stokes equations for each design. A state-of-the-art hierarchical-metamodel-assisted evolutionary algorithm has been used.

- A. Shake et al., 2014 discussed that on increasing of the runner velocity, the velocity decreases and the static pressure increases. The total recuperation of kinetic energy at the draft tube outlet is justified.
- B. Gunjan B. Bhatt et al. modeled draft tube using pro-program which reduces modeling time in the redesign. The CFD analysis has also been carried out for finding the pressure and velocity distribution, which matching with experimental results. Five different concepts for draft tube model have been proposed and compared to achieve highest efficiency and pressure at outlet.

Ruchi Khare et al. 2015 has discussed that flow distribution within draft tube depends on the amount of swirl and its direction coming out from runner. The flow becomes very uneven during the off-design operating conditions. This increases vibrations and need of flow to be uniform in draft tube. The splitter in draft tube can help to reduce the non-uniformity of flow. Elbow draft tube with splitter at different locations has been numerically simulated at rated conditions to study their effect on flow distribution in draft tube and its performance.

Sumeet J. Wadibhasme et al. 2016 have discussed about the principle of draft tube and its various types. Using the earlier work done by researchers, the parameter affecting the performance of draft tube are also discussed. The type of methods involved in the analysis of performance of draft tube are also considered.

Chakrabarty, S. et al. 2016 optimised the design of draft tube using ANSYS Fluent CFD code. The main objective of the research is to improve the efficiency by reducing the flow losses. Using the pressure and flow distribution, efficiency of the draft tube has been estimated for variation in length and height of draft tube.

Lekha Mourya et al. 2017 has emphasized that focus shall be given on Elbow Draft tube geometry. CFD should be used for analysis. Redesigning of the existing draft tube by changing their shapes has been studied.

Mun Chol Nam et al., 2018 used optimal Latin hypercube (OLH) method of the DOE technique for design optimisation of draft tube. Improvement in the performance is observed.

3. CONCLUDING REMARKS FROM LITERATURE REVIEW

Flow in draft tube affects the performance of reaction turbine runner. ANSYS CFX is the best software package for simulating flow and performing the detailed analysis. There is direct effect of length and height of draft tube on the overall performance of hydro electric power plant having reaction type turbine.

REFERENCES

1. Kubota, T., and S. Yamada. "Effect of Cone Angle at Draft Tube Inlet on Hydraulic Characteristic of Francis Turbine." Proc. IAHR 11th Symp., Amsterdam, Netherlands. No. 53. 1982.
2. Kubota, T., F. Han, and François Avellan. "Performance analysis of draft tube for gamm francis turbine." Hydraulic Machinery and Cavitation. Springer, Dordrecht, 1996. 130-139.

3. Ciocan, G. D., Iliescu, M. S., Vu, T. C., Nennemann, B., & Avellan, F. (2007). Experimental study and numerical simulation of the FLINDT draft tube rotating vortex. *Journal of Fluids Engineering*, 129(2), 146-158.
4. Iliescu, M. S., Ciocan, G. D., & Avellan, F. (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*, 130(2), 021105.
5. Altzibar, H., et al. "A draft-tube conical spouted bed for drying fine particles." *Drying Technology* 26.3 (2008): 308-314.
6. Gouin, Philippe, et al. "Experimental investigation of draft tube flow of an axial turbine by laser doppler velocimetry." *Proceedings of the 3rd IAHR International Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems*, Brno, Czech Republic, Oct. 2009.
7. Prasad, V., Khare, R., & Chincholikar, A. (2009). Numerical Simulation for Performance of Elbow Draft Tube at Different Geometric Configurations. *J. Continuum Mechanics, Fluids, Heat*, ISSN, 5095.
8. Khare, Ruchi., V. Prasad, and S. Kumar. "CFD approach for flow characteristics of hydraulic francis turbine." *International Journal of Engineering Science and Technology* 2.8 (2010): 3824-3831.
9. Susan-Resiga, Romeo, et al. "Analysis and prevention of vortex breakdown in the simplified discharge cone of a Francis turbine." *Journal of Fluids Engineering* 132.5 (2010): 051102.
10. Soni, Vishal, 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.
11. Brekke, H. (2010). Performance and safety of hydraulic turbines. In *IOP Conference Series: Earth and Environmental Science* (Vol. 12, No. 1, p. 012061). IOP Publishing.
12. Bhatt, G. B., Shah, D. B., & Patel, K. M. Design Modification of Draft Tube for Hydro Power Plant using CFD Analysis.
13. Makibar, J., et al. "Investigations on heat transfer and hydrodynamics under pyrolysis conditions of a pilot-plant draft tube conical spouted bed reactor." *Chemical Engineering and Processing: Process Intensification* 50.8 (2011): 790-798.
14. Khare, R., Prasad, V., & Mittal, S. K. (2012). Effect of runner solidity on performance of elbow draft tube. *Energy Procedia*, 14, 2054-2059.
15. Khare, R., Prasad, V., & Verma, M. (2012). Design Optimisation of conical draft tube of hydraulic turbine. *IJAEST International Journal of Advances in Engineering, Science and Technology*, 2(1), 22-26.
16. Rahul bajaj, Dr. Ruchi khare and Dr. Vishnu prasad , 2014. Effect of diffuser length on performance characteristics of elbow draft tube with dividing pier. *International Journal of Mechanical Engineering & Technology (IJMET)* - Volume:5,Issue:5,Pages:1-12
17. McNabb, J., Devals, C., Kyriacou, S. A., Murry, N., & Mullins, B. F. (2014). CFD based draft tube hydraulic design optimization. In *IOP Conference Series: Earth and Environmental Science* (Vol. 22, No. 1, p. 012023). IOP Publishing.
18. Siake, A., et al. "Hydrodynamic Characterization of Draft Tube Flow of a Hydraulic Turbine." *International Journal of Hydraulic Engineering* 3.4 (2014): 103-110.
19. Khare, R., Prasad, V., & Sharma, M. 2015. NUMERICAL SIMULATION FOR EFFECT OF SPLITTERS ON PERFORMANCE OF ELBOW DRAFT TUBE
20. Chakrabarty, S., Sarkar, B. K., & Maity, S. (2016, January). CFD Analysis of The Hydraulic Turbine Draft Tube to Improve System Efficiency. In *MATEC Web of Conferences* (Vol. 40). EDP Sciences.
21. Wadibhasme, S., Peshne, S., Barapatre, P., Barade, S., Dangore, S., Harde, S., & Daf, S. (2016). HYDRAULIC TURBINE DRAFT TUBE: LITERATURE REVIEW. *International Journal of Science, Engineering and Technology Research (IJSETR)*, 5(3).
22. Lekha Mourya, Rahul Mishra, Swapnil Jain. 2017. A review of literature on elbow draft tube geometry of kaplan turbine. Volume 5 Issue VI, 2097-2101.
23. Chol Nam, Mun, et al. "Design optimization of hydraulic turbine draft tube based on CFD and DOE method." *IOP Conference Series: Earth and*

Environmental Science. Vol. 136. No. 1. IOP Publishing, 2018

24. ANSYS CFX 12 and 13 software manuals.
25. Kumar D.S., "Fluid Mechanics and Fluid Power Engineering", (2008), S. K. Kataria and Sons, Delhi.
26. Modi P.N., Seth, S.M., "Hydraulics and Fluid Mechanics including Hydraulic Machines", (2011), Standard Book House, Delhi.