REVIEW ON THE ADVANCES IN DESIGN OF DRAFT TUBES FOR REACTION TURBINES

Sunita Mohanty[1], Prof. Akshay Anand[2]

[1]M.Tech. Scholar, Millennium Institute of Technology, Bhopal, India.

[2]Assistant Professor (M.E.), Millennium Institute of Technology, Bhopal, India.

ABSTRACT

Energy is available in various forms in nature. Electricity is the most efficient form of energy that can be transmitted and used by humans.

To produce electricity, one of the ways is to harness water of flowing water using reaction turbines. Efficiency of turbines depends upon performance of every part of the machine. In the present paper advances in the design of hydraulic turbines in the recent years has been studied. It can be concluded that Draft Tube is one of the important Part of Reaction Turbines and its' efficiency plays a vital role. There is a scope in design parameters improvement of the draft tube.

1. INTRODUCTION

In hydro power plants involving reaction type turbines, draft tube is a component commonly used. The primary function of the draft tube is to improve the efficiency and performance of hydraulic machines by helping to control the flow of fluid. When fluid (such as water) passes through a reaction type hydraulic turbine, it gains kinetic energy due to the machine's action.

In turbines, this kinetic energy is converted into mechanical energy that drives a generator to produce electricity. However, after passing through the turbine, the fluid often has high velocity and low pressure, which can lead to energy losses and undesirable effects. The purpose of the draft tube is to recover some of the kinetic energy in the fluid leaving the turbine. Draft tubes come in various shapes and designs, such as straight, conical, or diffuser-shaped, depending on the specific application and the design requirements of the machine. The shape and dimensions of the draft tube are critical to its performance, as they determine how effectively pressure recovery and flow stabilization are achieved.

With numerical techniques and computational power, Computational Fluid Dynamics (CFD) has emerged as a boon. CFD can be used to find out the performance of the hydraulic components. A lot of work and studies have been done on turbines and its hydraulic components using numerical methods involving CFD which are described in various research papers. Few of the papers which are referred are briefly described below.

2. LITERATURE REVIEW

Many researchers have worked for design optimization of various hydraulic components. CFD is one of the promising tools for performing the same. Few noteworthy contributions are:

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

Prasad, V.et al. 2009 numerically carried out 3D viscous turbulent flow analysis of an elbow type draft tube. The draft tube's height and length were adjusted together with the mass flow rates. The code used was ANSYS CFX. Based on pressure and velocity distributions, efficiencies and losses are calculated for the draft tube and graphically displayed. Obtained geometry for best performance is very similar to the geometry employed in the majority of hydroelectric power plants.

Rahul Bajaj et al. 2014 found that the geometry of draft tube had large impact on the performance of reaction turbine. Using CFD, alternative designs of draft tube have been investigated. A huge elbow draught tube with an integrated pier diffuser has undergone numerical simulation. To examine the impact of the alteration on the effectiveness of the draft tube, the diffuser's length was altered.

Gunjan B. Bhatt et al. modeled draft tube using pro-program which reduces modeling time in the redesign. Additionally, the CFD analysis was done to determine the pressure and velocity distributions that matched the outcomes of the experiments. For the draft tube model, five distinct ideas have been put forth and contrasted in order to attain the best efficiency and outlet pressure.

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. In order to gain a physical understanding of the complicated hydrodynamic dynamics involved in the formation of the cavitation rope, the impact of the turbine setting level on the volume of the cavity rope has been studied.

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 simulation results were used to compute the draft tube's performance characteristics, and the impact of runner solidity and rotational speed on that performance was explored. The comparison of numerical results with experimental results shows a good agreement.

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

Ruchi Khare et al. 2012 has stated that because conical draft tubes recover more of the vortex flow leaving the runner than elbow draught tubes, they perform more efficiently. Conical draft tubes have been numerically simulated using ANSYS CFX 13.0 to account for variations in length and diffuser angle. By deriving head loss, head recovery coefficients, and efficiency of draught tube from simulation data, the performance of the draught tube has been examined.

Sumeet J. Wadibhasme et al. 2016 talked about the many sorts and the draft tube's basic idea. The parameters affecting the performance of the draft tube are also examined using past research. It is also taken into account what procedures are used to analyse the performance of the draft tube.

Chakrabarty, S. et al. 2016 used the ANSYS Fluent CFD code to optimise the design of the draft tube. The research's primary goal is to increase efficiency by lowering flow losses. The efficiency of the draft tube has been calculated for variations in length and height using pressure and flow distribution.

McNabb, J., et al. 2014 described a method for resolving this optimisation issue. A complete set of design parameters is used to define the precise geometry of the draft tube. For each design, analysis was performed using the Navier-Stokes equations. Modern evolutionary algorithms with hierarchical metamodel assistance have been deployed for the same.

Ruchi Khare et al. 2015 has explored how the amount and direction of swirl originating from the runner affects how the flow is distributed inside the draft tube. In the off-design operating conditions, the flow becomes incredibly uneven. This causes more vibrations and a need for a consistent flow in the draft tube. The splitter in the draft tube can aid in minimising the irregularities in the flow. To investigate their impact on the performance of the draft tube and the flow distribution in it, various positions of the elbow draft tube with splitters have been numerically simulated at rated circumstances.

Brekke, H. 2010 has talked about picking tiny turbines for run-of-river power projects. Along with safety and necessary maintenance of turbines, with a focus on bolt connections, optimisation of the performance of various types of large turbines has also been covered.

Gupta et al. 2015 studied that hydraulic turbines are made to operate as efficiently as possible. Each component of the hydraulic electric power plant must operate at close to 100% efficiency in order to achieve the best efficiency feasible. Draft tubes are a key component of reaction turbines and are used to lower discharge velocity while increasing discharge pressure.

The efficiency of the draught tube also plays a significant role in the design of the response turbine because its performance directly impacts the output of the turbine as a whole. The draught tube's performance is influenced by its size, diffuser angle, and shape. The draft tube efficiencies and losses are computed from pressure and velocity values obtained from CFD Post at inlet and outlet. Results are also presented graphically to visualize energy conversion in draft tube.

3. CONCLUSION

Efficiency of draft tube plays a very important role in the overall performance of reaction turbines. Numerical methods can be used to predict the performance of hydraulic components in design stage only.

ANSYS CFX is one of the promising software packages for simulating flow and performing the detailed analysis of hydraulic components. There is direct effect of length and divergent angle of draft tube on the overall performance of hydro electric power plant having reaction type turbine.

4. 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] 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.

[3] 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.

[4] 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.

[5] 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.

[6] 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.

[7] Khan, M. H., Tiwari, M. K., & Gupta, V. PREDICTION OF EFFICIENCY OF CONICAL DRAFT TUBE USING NUMERICAL METHOD.

[8] Gupta, V., Prasad, V., & Khare, R. Numerical Simulation For Visualising Effect Of Jet Shape On Various Parameters Of Multi Jet Pelton Turbine Model.

[9] Gupta, V., Prasad, V., & Khare, R. (2016). Numerical simulation of six jet Pelton turbine model. Energy, 104, 24-32.

[10] ANSYS CFX 15 software manuals.