Journal of Advanced Mechanical Sciences (JAMS)
ISSN: 2583-3790 (Online)
"J. Adv. Mech. Sci., vol. 1, no. 1, pp. 31-35, March 2022"

CFD Analysis in Francis Turbine Performance- A Review

Akash Gupta[a]*
[a]*PG Scholar, Thermal Power Engineering, National Institute of Technology, Tiruchirappalli, India.*
**Corresponding Author Email: ag012024@gmail.com,*
Article received: 03/03/2022, Article Revised: 27/03/2022, Article Accepted: 29/03/2022
Doi: 10.5281/zenodo.6482160

Abstract

In any hydro power plant its turbine enacts a major role in deciding the competence as lot of phenomena depends on it if turbine works smoother then performance achieved by plant will be more and for that analysis of turbine becomes a need of the time and this need is fulfilled by the CFD it offers various software packages to analyse the part numerically at virtually no cost and in this study various parameters affecting different component of Francis turbine are analysed using CFD in addition to some validation of CFD results with actual one are also mentioned.

Keywords: *CFD, Francis turbine, Hydro power plant, Cavitation, ANSYS.*

1. INTRODUCTION

Francis turbine is an outward flow reaction turbine and is used for production of electricity in hydro power plant its efficiency is quite more around 90% actually Francis turbine is not fully reaction turbine as its blades rotate under both impulse and reaction component. Firstly water enters through spiral casing which is made in such a way that its diameter decreases along flow to ensure constant velocity of water then fixed vanes is there to ensure water enters the guide blades without swirling and then from guide vanes we can select the optimum angle of attack to strike water on runner blades to get maximum power and efficiency. Since components used in Francis turbine are very costly and they are supposed to operate under different load condition and other parameters like temperature, pressure drop, corrosion so before making the parts has to be analysed and no one supposed to do it experimentally because of the cost involved hence here CFD offers tools to analyse the components without making it actually then with proper tools and simulations we can get the optimum value of parameters involved and even can get the condition under which parts are supposed to fail and parameters where it can give maximum performance.

2. EARLIER STUDY OF FRANCIS TURBINE USING CFD:

Navthar et al.[1] tries to use CFD to predict the failure and also to find optimum performance parameters with help of simulation and also by choosing appropriate boundary condition at inlet and outlet they gets variation of pressure, velocity contour along the blade profile which make them able to analyse Francis turbine in few time and effectively without any cost as related to experimentation method.

Tiwari et al. [2] performed a work to understand the need of CFD in both strategy as well as performance stage for a general hydraulic turbine in report they analyse operating characteristics, unsteady phenomena, analysis of cavitation, losses, and importance of boundary condition and turbulence modelling in CFD and objective of analysis is turbine should operate at optimum efficiency under different operating condition they found that since cavitation also effect performance and its very cumbersome to analyse so the importance of boundary and initial condition in addition to turbulence modelling is always there to get the accurate design condition so that cavitation must be predicted accurately.

Zeng et al.[3] uses CFD to analyse unsteady flow in draft tube and for that they uses equation of Navier –Stokes and large eddy simulation(LES) for simulation of vortex flow involved they uses fluent software and they find that their result confirm to experimental method hence we can predict the zone of stable operating condition which is important in performance and design of turbine using numerical method.

Choi et al. [4] had to replace turbine and for that they uses CFD to increase and predict the performance of 500 kW Francis turbine also for that they uses CFD Ansys package and their CFD based design model is able to upgrade the peak efficiency around 10% and for that they have done turbulent flow simulation, coupled calculation also at design and off load condition. Moreover they even get smooth range of operation as cavitation gets reduced in new design.

Krishna et al. [5] worked on design of Francis turbine blade in order to get minimum sediment erosion while maintaining efficiency using CFD .there is considerable reduction in efficiency due to silt erosion with the use of CFD simulation they infer that erosion rate decreases around 31 times when there is 25% curvature in profile blade at the same time reduction is efficiency is limited to 0.25% also it can be seen that erosion increases and efficiency started decreases when outlet angle increases beyond 20^o.

Teran et al. [6] try to improve geometry of components for that they uses artificial neural network, genetic algorithm and experience of existing worker improvement suggested using the results of validation but this only applied to geometries other than blade profile and then artificial neural network, genetic algorithm is used for improvement of runner blade profile. After all these efficiency of turbine increases around 15% at maximum power and at optimum flow rate around 16% energy production increased in compare to existing one.

Gohiland Saini [7] analyses the unsteady cavitation in a 200kW Francis turbine prototype basically at three different condition namely rated load, part load and overload condition they used the CFX code under transient condition and shear stress transport model they come to conclusion that minimum loss in efficiency is come out to be at rated load and maximum at overload condition as in overload condition.

Santiago et al. [8] performed simulation of a flow inside Francis turbine numerically out of four simulation methodology namely Hill chart determination, Losses analysis (efficiencies), Rotor-stator interaction analysis and Pressure fluctuations they worked on first and third one and it has been found that for both hill chart i.e. steady analysis and rotor stator interaction i.e. unsteady analysis numerical results shows good with experimental setup.

Arispe et al. [9] study a reduced model of Francis turbine for draft tube parameterization which finds its importance in draft tube outlet to recover part of pressure energy four type of geometry are considered for draft tube and ANSYS CFX was used for numerical simulation and found that it has been found that draft tube with hyperbolic shape and curve with logarithmic spiral shape have higher efficiency than original geometry and the other.

Manoj et al.[10] validate the result with experimental one for 3 D flow analysis in Francis turbine using ANSYS CFX 11 simulation analysis has been done and it has been found that losses come out to be minimum at best operating condition but at the peak condition there is difference in result between experimental and computational also there is slight difference in efficiency too and all these may supposed to occur in discretizing the governing equation and somewhat instrumental and human mistake.

Hasan et al.[11] discusses design methodology of Francis turbine using CFD firstly components like spiral case was designed using Matlab code then it is modelled with help of CAD after that it is redesigned using CFD that flow become uniformly distributed and for the analysis part CFD v.11 CFX solver was used similarly all components are analysed using

CFD and presented in the end proposed methodology was used in actual hydro power plant and overall efficiency come out to be around 92%.

Sanjay et al. [12] consider 3 MW capacity power plant to predict its efficiency using CFD approach for that and with use of commercial CFD package simulation were performed and two boundary condition were taken namely i. inlet and outlet pressure and ii. Mass flow inlet and pressure outlet it is found that out of these two second boundary condition were optimum one. Semerci et al. [13] analyse the performance of turbine using CFD codes by varying vane position and for each position efficiency is compared to find out the optimum position at which it gives better performance. Five separate CFD analyse has been carried out to get optimum angle of guide vane position and around 22^{o} efficiency come out to be maximum.

Wahidullah et al. [14] analyses Francis turbine performance using CFD in order to get flow behaviour and losses incurred they found with varying rotational speed pressure and velocity variation are similar but with guide vane angle they varies considerably also maximum efficiency and power came out to be at same optimum rotational speed and pressure and streamline contour of different confirms to actual flow in turbine.

Ayancik et al.[15] analyses the runner parameters on design of Francis turbine and how it affect the performance from CFD a methodology is being implemented to find the best efficiency and with that new model of runner is generated using CFD results and it is found that by changing flow and blade angle blade shape is being determined.

3. CONCLUSION

- To get accurate result from CFD it is found that initial and boundary condition plays a vital role especially in analysingcavitation.
- CFD based design model to replace existing model can alsobe implemented successfully.
- Shape of draft tube also needs to be analysed to predict the performance as it plays a vital role.
- Validation of CFD results with actual testing results conforms most of time slight difference may be there due to human error and improper discretization.
- Guide vane optimum angle runner blade parameters also decides the performance of turbine so simulation has been done for this part too by researchers.

Acknowledgement/Funding Acknowledgement

The author(s) received no financial support for the research, authorship, and/or publication of this article.

Declaration of Competing Interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

REFERENCES

[1] Navthar, R. R., TejasPrasad, J., Saurabh, D., Nitish, D., & Anand, A. (2012). CFD analysis of Francis turbine. *International Journal of Engineering Science and Technology*, *4*(7), 3194-3199.

[2] Tiwari, G., Kumar, J., Prasad, V., & Patel, V. K. (2020). Utility of CFD in the design and performance analysis of hydraulic turbines—A review. *Energy Reports*, *6*, 2410-2429.

[3] Zeng, Y., Liu, X., & Wang, H. (2012). Prediction and experimental verification of vortex flow in draft tube of Francis turbine based on CFD. *Procedia Engineering*, *31*, 196-205.

[4] Choi, H. J., Zullah, M. A., Roh, H. W., Ha, P. S., Oh, S. Y., & Lee, Y. H. (2013). CFD validation of performance improvement of a 500 kW Francis turbine. *Renewable Energy*, *54*, 111-123.

[5] Khanal, K., Neopane, H. P., Rai, S., Thapa, M., Bhatt, S., & Shrestha, R. (2016). A methodology for designing Francis runner blade to find minimum sediment erosion using CFD. *Renewable Energy*, *87*, 307-316.

[6] Teran, L. A., Larrahondo, F. J., & Rodríguez, S. A. (2016). Performance improvement of a 500-kW Francis turbine based on CFD. *Renewable energy*, *96*, 977-992.

[7] Gohil, P. P., & Saini, R. P. (2015). Numerical study of cavitation in Francis turbine of a small hydro power plant. *Journal of Applied Fluid Mechanics*, *9*(1), 357-365.

[8] Laín-Beatove, S., Ruiz, M. J. G., Quintero-Arboleda, B., & Orrego-Bustamante, S. (2010). CFD numerical simulations of francis turbines. *Revista Facultad de Ingeniería Universidad de Antioquia*, (51), 31-40.

[9] Laín-Beatove, S., Ruiz, M. J. G., Quintero-Arboleda, B., & Orrego-Bustamante, S. (2010). CFD numerical simulations of francis turbines. *Revista Facultad de Ingeniería Universidad de Antioquia*, (51), 31-40.

[10] Shukla, M. K., Jain, R., Prasad, V., & Shukla, S. N. (2011). CFD analysis of 3-D flow for francis turbine. *MIT International Journal of Mechanical Engineering*, *1*(2), 93-100.

[11] Akin, H., Aytac, Z., Ayancik, F., Ozkaya, E., Arioz, E., Celebioglu, K., & Aradag, S. (2013, May). A CFD aided hydraulic turbine design methodology applied to Francis turbines. In *4th International Conference on Power Engineering, Energy and Electrical Drives* (pp. 694-699). IEEE.

[12] Jain, S., Saini, R. P., & Kumar, A. (2010, October). CFD approach for prediction of efficiency of Francis turbine. In *The 8th International Conference on Hydraulic EfŸiciency Measurement*.

[13] Semerci, D.S., & Yavuz, T. (2020, December 2020). CFD-Based Performance Analyses of a Francis Turbine in Several Guide Vane Positions. *Conference ICAT*.

[14] Safi, W. H., & Prasad, V. (2017). Design and permance analysis of Francis turbine for hydro power station on Kunar river using CFD. *International Journal of Advanced Research*, *5*(5), 1004-1012.

[15] Ayancik, F., Celebioglu, K., & Aradag, S. (2014). Parametrical and theoretical design of a Francis turbine runner with the help of computational fluid dynamics. *International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics*.