Journal of Basic and Applied Engineering Research
p-ISSN: 2350-0077; e-ISSN: 2350-0255; Volume 3, Issue 5; April-June, 2016, pp. 466-467

http://www.krishisanskriti.org/Publication.html

Design, Analysis and Optimization of a Runner Blade in Small Bulb Turbine

Mahesh[1] and Kumar Sambhav[2]

[1]M.Tech. Student, CAD/CAM, School of Mechanical Engineering, Galgotias University
[2]School of Mechanical Engineering, Galgotias University
E-mail: [1]maheshnadda51@gmail.com,[2]kumar.sambhav@galgotiasuniversity.edu.in

Abstract—*Hydro-power generation through flowing water in the rivers is a good alternative to making big dams. Bulb turbines offer the potential to tap the energy in the flowing water for power generation. In this paper, the runner blades of a small bulb turbine have been optimized at different pitch angles for maximum power generation using CFD analysis. The turbine blades, hub and inlet domain are modeled at different pitch angles for a given velocity of water, and the impact of varied pitch angles is evaluated on the power output, torque and turbine efficiency. First a large range of pitch angles is selected, and slowly the solution is allowed to converge. An SST k-ω turbulence model is used during CFD analysis in ANSYS software.*

Keywords*: Bulb turbine, CFD, performance prediction, SST k- ω turbulence model, pitch angles.*

1. INTRODUCTION

Horizontal bulb turbine is widely used in small hydropower because they are appropriate to be used low head of hydropower resources. Runner design is one of the most important tasks that manufacturers carry out when optimizing a bulb unit. In modern hydraulic turbine design, improving the efficiency, reducing cavitation factor and enhancing the stability of the unit has become more and more important. Turbine blade is the core flow component of whole flow passes and play the most important role in energy conservation. So design optimization of turbine runner is the key issue to obtain a turbine with desired performance. Mainly laboratory experiments were carried using physical model to know the hydrodynamic behavior of the machine. A CFD based 3D CAD analysis technique replaces costly experimental procedures by theoretical analysis. This technique is time and cost saving [1,2]. Reynolds averaged Navier-Stoke(RANS)equation in combination with Boussinesq hypothesis [2], which assumes isotropic eddy viscosity.

Many factors affect performance of bulb turbine like design of blade, thickness of the leading edge

(which is thicker than trailing edge for a streamlined flow, Timo Flashopler [7] and thickness of blade (which should be as thin as possible) to improve cavitation characteristics. Axial flow of water, with a component of swirl, applies force on blades of rotor and loses its momentum, both linear and angular, producing torque and rotation in shaft which is connected to generator. In this paper an optimization method of runner blades based on CFD analysis is built. The hydraulic performance can be improved through different blade pitch angles. At individual pitch angles, CFD analysis is done and hydraulic performance is analyzed. For optimization, this process has been repeated for different pitch angles and optimized pitch angle is achieved.

2. METHOD OF APPROACH

The aim of present work is to find relationship between blade pitch angle and performance of bulb turbine at the water velocity of 2 m/s. The diameter of runner is 1m. The dimensions of bulb turbine are showing in Fig. (1). The number of blades is chosen as three and the shape of blade is aerodynamic in nature. The diameter of hub is 250 mm and diameter of outlet shroud is greater than inlet.

A 3D CAD model is generated and numerical flow simulation is performed for a number of operating conditions. Runner blade is set at different pitch angles and performance is analyzed for maximum output power. Optimization is done for maximum output power for a pitch angle. The blade profile is generated and blade pitch angle is setup in ANSYS BladeGen tool and CFD analysis is done on ANSYS CFX 14.5 software.

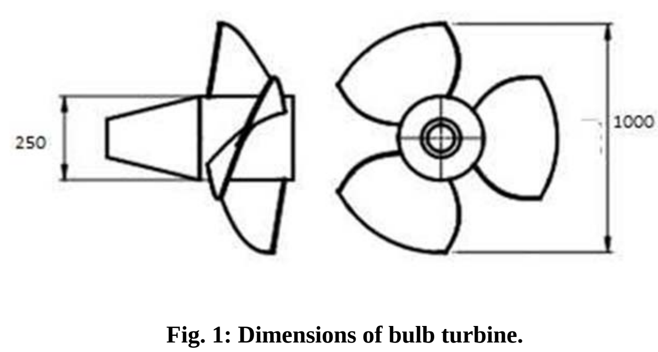

Fig. 1: Dimensions of bulb turbine.

3. NUMERICAL MODEL

The incompressible continuity equation and the Reynolds time average Navier-Stokes equation with SST k-ω turbulence model was adopted to simulate through turbine passage. The particular implementation of the SST k-ω model has been described in detail by Menter [4] who also gave very accurate prediction of the onset and amount flow separation under adverse pressure gradients to account for the turbulent shear stress. A 3D CAD model is showing in Fig. (2) and meshing is done with ANSYS TurboGen tool. Meshing type is hexahedra with number of nodes 156200 and elements 144436 as shown in Fig. (3). To capture very small features of turbine near the wall yplus value is given between 0 to 1249.95. Interfaces between rotating and non-rotating components of the machine are specified based on multiple frame reference concept. Frame change metohds available in CFX are frozen rotor, stage interface and transient rotor stator. Among these, stage interface is one of the simplest methods and this has been used for modeling. This is the method of circumferential averaging where flow variables at the interface are averaged in a circumferential direction and solved for the given boundary conditions.

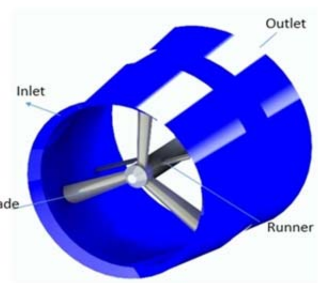

Fig. (2) 3D CAD model of turbine

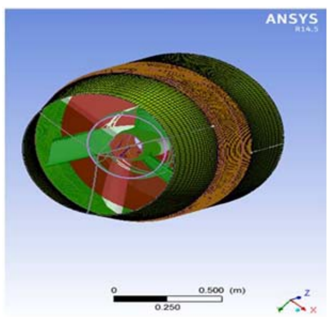

Fig. (3) Meshing of Turbine

4. BOUNDARY CONDITIONS AND CONVERGENCE

Boundary condition represents the known computational values within the ends of the spatial domain for any temporal variations. The transient type of analysis is done for 80 time steps for inlet boundary condition given input velocity of water 2m/s. at 1 atm pressure. The main components of bulb turbine are defined as blade, hub, inlet, outlet, shroud and their interfaces. These boundary conditions are fed in ANSYS CFX Turbo tool with their given value and boundary specification.

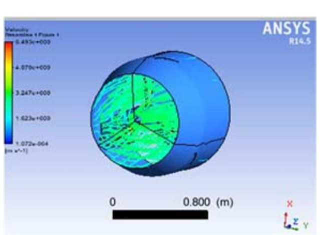

Fig. 4: Velocity profile at stagger angle of 350

Fig. 5: Graph between pitch angle and torque

5. SIMULATION RESULTS AND DISCUSSION

The calculation was performed for different pitch angles 20^0, 30^0, 35^0, and 45^0. First numerical simulation is applied to the original bulb turbine and results are compare d with different pitch angle with the help of their output torque. The optimum value of pitch angle is obtained as 35^0. The velocity profile for the same is shown in Fig. 4, and the torque value at this angle is 2647.03 N-m as shown in Fig. 5. By using this methodology, we can predict the performance of turbine and optimize for maximum output power for any given R.P.M.

6. CONCLUSION

The presented work establishes the optimum pitch angle for a bulb turbine with runner diameter of 1m in flowing water with a speed of 2 m/s as 35^0. In order to obtain this optimum value, a floating bulb turbine was modeled in Ansys14.5 and an SST k-ω turbulence model was used for CFD analysis. Such optimum designs of floating type bulb turbine can utilize flowing river water energy, which is a renewable energy source, to fulfill the energy needs of rural areas of the country in a cost-effective and sustainable manner.

REFERENCES

[1] Benišek, M., Čantrak, S., Nedeljković, M., Ilić, D., Božić, I. and Čantrak, Đ.: Defining the optimum shape of the cross-flow turbine semi-spiral case by the lagrange's principle of virtual work, FM Transactions, Vol. 33, No. 3, pp. 141-144, 2005.

[2] Vu, T.C., Koller, M., Gauthier, M. and Deschênes, C.: Flow simulation and efficiency hill chart prediction for a Propeller turbine, 25th IAHR Symposium of Hydraulic Machinery and Systems, Timiosara, Romania, September 20-24, 2010.

[3] Wilcox, D.: Turbulence Modeling for CFD, DCW Industries Inc. La Canada, 1994.

[4] Jots, D., Lipej, A., Oberdank, K., Jamnik, M. and Velenšek, B.: Numerical flow analysis of a Kaplan turbine, Proceedings of the XVIII IAHR Symposium on Hydraulic Machinery and Cavitation, Volume II, pp. 1123-1132, 1996.

[5] Brost, V., A. Ruprecht, and M. Maihöfer: Rotor-stator Interactions in an Axial Tur-bine, A Comparison of Transient and Steady State Frozen Rotor Simulations. In Conference on Case Studies in Hydraulic Systems-CSHS03, Belgrade, 2003

[6] F R Menter: Two equation eddy- viscosity turbulence models for engineering applications, ,1994

[7] Liu, ShuHong, Jie Shao, ShangFeng Wu, and YuLin Wu. 2008. "Numerical Simulation of Pressure Fluctuation in Kaplan Turbine." Science in China Series E: Technological Sciences 51 (8) (July 6): 1137– 1148. doi:10.1007/s11431-008-0159-9.

[8] Nilsson, H., and L. Davidson. 2003. "Validations of CFD Against Detailed Velocity and Pressure Measurements in Water Turbine Runner Flow." International Journal for Numerical Methods in Fluids 41 (8): 863–879.

[9] Anagnostopoulos, J. S. and Papantonis, D. E. 2007." Flow Modeling and Runner Design Optimization in Turbo Water Turbines", World Academy of Science, Engineering and Technology, 206211.

[10] [25] Chen HC, Patel VC. Near-wall turbulence models for complex flows includingseparation. AIAA J 1988;26(6):641–8.