INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH & DEVELOPMENT

Unsteady Simulation Of A Vertical Axis Wind Turbine

Sukanta Roy
Department of Mechanical Engineering, Indian Institute of Technology Guwahati, Guwahati, India
Keshab Biswakarma
Department of Mechanical Engineering, Royal School of Engineering and Technology, Guwahati, India
Ujjwal K. Saha
Department of Mechanical Engineering, Indian Institute of Technology Guwahati, Guwahati, India

Abstract:
In the present study, an unsteady computational investigation has been carried out in order to analyze the performance of a single-stage vertical axis wind turbine using ANSYS Fluent 6.3 software package and the results obtained are validated with the experimental data. A Realizable $k-\varepsilon$ *turbulence model with enhanced wall functions has been used to analyze a three dimensional model of this turbine. The data and boundary conditions for this analysis have been taken from the experimental observations. The experiments have been carried out under various load conditions to find the power and torque coefficients. The outcomes of numerical simulation and experiments are compared, analyzed and are found analogous with the data available in open literature.*

Keywords: *Vertical axis wind turbine, power coefficient, computational fluid dynamics*

Introduction

The vertical axis wind turbines (VAWTs) are being popular in the renewable energy field to harvest energy from the wind for small scale applications because of its design simplicity, better self-starting capability, lower installation and maintenance cost and direction independency [1]. In order to design an efficient and high performance VAWT, it is important to have a good understanding of how the design and the operating parameters affect the flow fields around the turbine. Many researchers to elucidate the power mechanism of the rotor have investigated the flow field around a turbine, which is closely related to its torque and power performance. Due to advancement of general purpose computational fluid dynamics (CFD) software packages, CFD has turned out to be a powerful tool to complement the expensive experimental findings in flow structure studies such as flow over a VAWT [2, 3]. In the present investigation, a three dimensional computation study has been made on a 3 bladed single stage vertical axis wind turbine with the help of ANSYS Fluent 6.3 software package and Pro-E software. The experiments have been carried out to study the effect of load on the rotor performance. In order to study the performance characteristics of this turbine, the torque and power coefficient data obtained from both the studies are compared and analyzed.

Computational Methodology

The modeling of the turbine blades has been done with the help of Pro-E software and ACIS file has been exported, which is then imported to Gambit software for complete 3-D modeling as shown in Fig. 1. The schematic diagram of the single stage VAWT is shown in Fig. 2. The complete computational model includes an inner cylinder containing the turbine model and an outer box containing another hollow cylinder, which exactly fits the inner cylinder and turbine model. The swept area of the turbine is 1.836 m^2. The width, breadth and height of the computational domain of the outer box are taken as 2.5 m each. The meshing is done in Gambit using a Tet/Hybrid element and TGrid type as shown in Fig. 3. A total of 1600000 cells have been taken after the mesh refinement. The boundary conditions given to the three dimensional turbine model are as follows: velocity inlet, pressure outlet, both the cylinders as interface and turbine blades as moving wall as shown in Fig. 4.

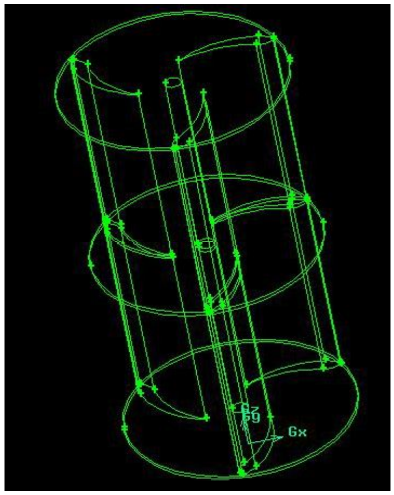

Figure1: 3D turbine model in Gambit

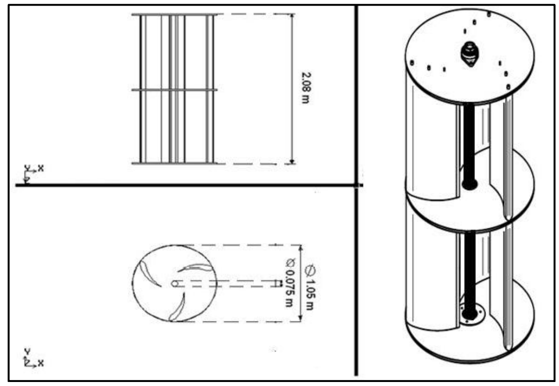

Figure2: Schematic diagram of the 3-bladed VAWT

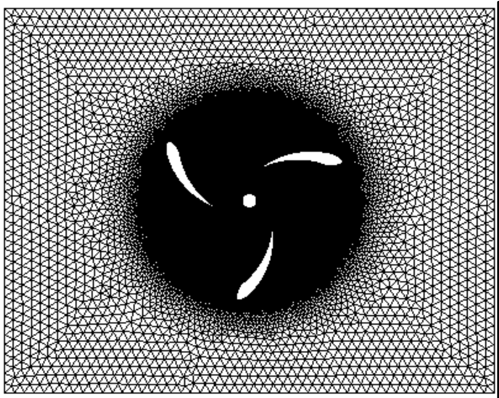

Figure 3: Meshing of the model in Gambit.

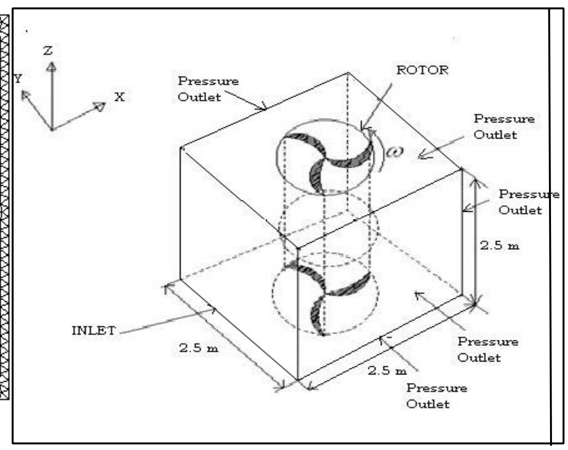

Figure 4: Boundary condition of the 3D model.

Once the Gambit model has been prepared, the 3D mesh has been exported and analyzed using the CFD-Fluent software. In the flow analysis, the convergence criterion to solve the Navier-Stokes equation is taken as 10^{-3}, whereas 0.001 sec time step size and 20 iterations per time step are taken for the iteration. The simulation is carried out for 50000 time steps in order to have a number of complete rotations of the turbine model. The following assumptions are made to select the model and solver for this analysis: 3D unsteady pressure based solver with 2nd order implicit formulation, Realizable $k-\varepsilon$ turbulence model with enhanced wall functions and SIMPLE pressure-velocity coupling with 2nd order upwind discretization. The transport equations for Realizable $k-\varepsilon$ turbulence are:

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+(G_k+G_b)-\rho\varepsilon+S_k \quad (1)$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial}{\partial x_i}(\rho\varepsilon u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+\rho C_1 S\varepsilon+C_{1\varepsilon}\frac{\varepsilon}{k}C_{3\varepsilon}G_b-C_2\rho\frac{\varepsilon^2}{k+\sqrt{v\varepsilon}}+S_\varepsilon \quad (2)$$

where, $C_1 = Max\left[0.043, \frac{\eta}{\eta+5}\right]$, $\eta = S\frac{k}{\varepsilon}, S=\sqrt{2S_{ij}S_{ij}}$, G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients, G_b is the generation of the turbulence kinetic energy due to buoyancy.

The Realizable $k-\varepsilon$ turbulence model has been extensively validated for a wide range of flows, including rotating homogeneous shear flows, free flows including jets and mixing layers, channel and boundary layer flows, and separated flows. For all these cases, the performance of this model has been found to be substantially better than that of standard $k-\varepsilon$ model [3]. Looking at the advantages, the Realizable $k-\varepsilon$ model has been chosen for this computational study. For dynamic meshing in the ANSYS-Fluent software, turbine blades have been assumed as moving wall rotating with adjacent cells and rotating fluid (inner cylinder) as moving mesh. The incoming velocity is considered as 5.785 m/s, the mass flux average velocity of all the velocities obtained from the experiments at different locations of the turbine. The turbine is connected with a three phase alternator through a pulley drive having a ratio of 1:4. The alternator output is connected to different bulb loads. By increasing the load, the power and the torque values have been calculated.

Results And Discussion

From the experiments, the angular velocity of the turbine at no load is found to be 9.215 rad/s. From the linear extrapolation of the experimental data, the torque obtained at no-load condition has been found to be 4.72 N-m at a turbine speed of 88 rpm. At a tip speed ratio, μ = 0.836, the torque coefficient (C_m) and the power coefficient (C_p) at no-load condition are found to be 0.239 and 0.172, respectively. Figure 5 shows the effect of increasing the bulb-load on the torque data at an average wind speed of 5.785 m/s. It has been observed that with the increase of bulb load the turbine shaft speed decreases at a constant wind speed.

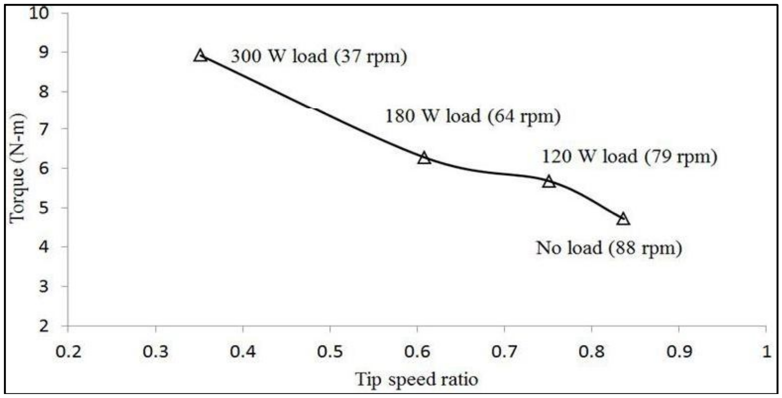

Figure 5: The effect of load on torque data for single stage VAWT at wind speed 5.785 m/s

From the simulation, the velocity and the static pressure contours are plotted in the Figs. 6 and 7 respectively. These show a high static pressure on the returning blade of the turbine and high velocity magnitude near the tip of the turbine blades. This high pressure on the returning blade is responsible for low net torque value of this turbine. The mean torque coefficient value has been calculated by averaging the torque coefficient at different time steps and hence, the mean torque value for this turbine has been calculated. Figure 8 shows the torque coefficients at different time intervals for unsteady simulation of three dimensional VAWT. Figure 9 shows the torque values at different time intervals for the unsteady simulation at a mean wind velocity of 5.785 m/s. Finally, the mean torque coefficient (0.223), mean torque (4.41 N-m); power (40.63 W) and power coefficient (0.16) for this single stage turbine has been calculated from this computational study. These computational and experimental results match well with the data available in published literature where the power coefficient lies between 0.15-0.18 [4, 5].

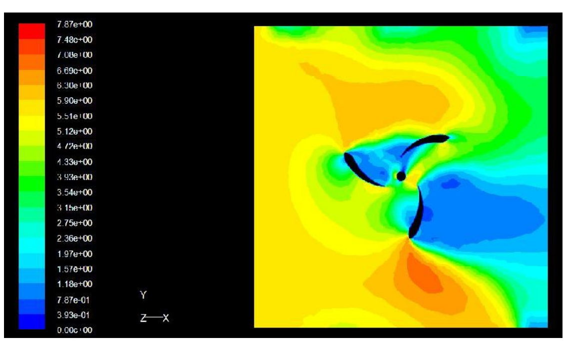

Figure6:Velocity magnitude contours of turbine.

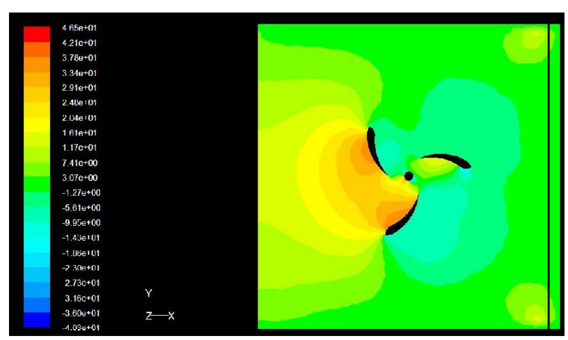

Figure7:Static pressure contours of turbine.

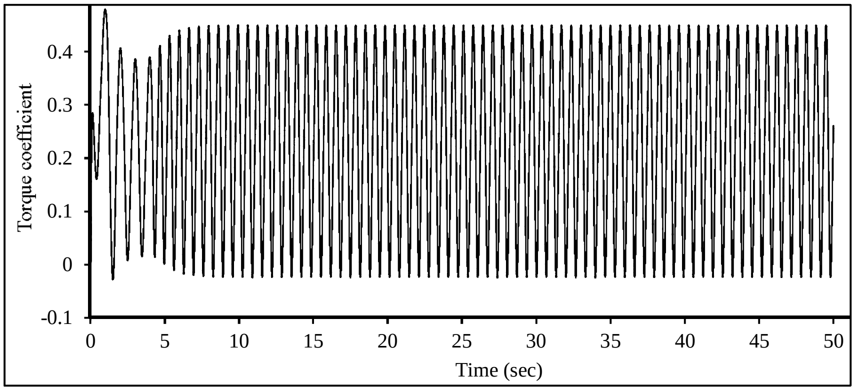

Figure 8: Torque coefficients for unsteady CFD simulation of 3-bladed single stage VAWT

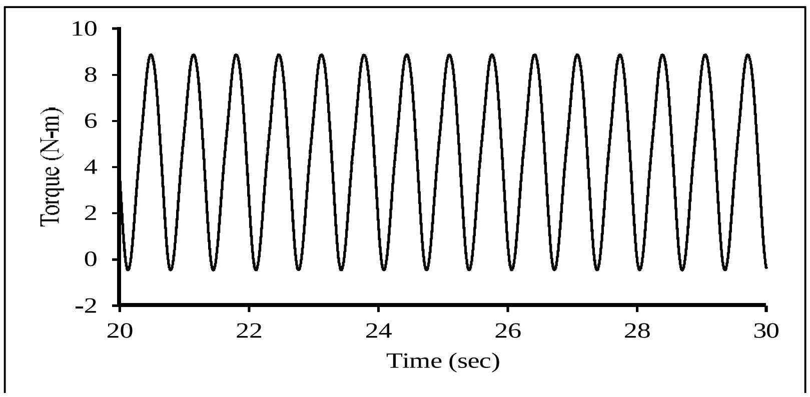

Figure 9: Torque value for unsteady CFD simulation of 3-bladed single stage VAWT

Conclusion

The present three dimensional computational study guaranties the use of CFD Fluent software for the flow analysis of wind turbines. For this single stage turbine, the experimental results have shown a power coefficient of 0.172 and a torque coefficient of 0.239 at a tip speed ratio of 0.836. On the other hand, the computational results have shown a power coefficient of 0.16 and a torque coefficient of 0.223. These results have also shown a good agreement with the results available in the literature [4, 5]. The computational results are within 7% deviation from the experimental results, which encourages the further research scope for CFD to analyze the flow dynamics over the VAWT and improve its design and performance.

Reference

1. Tabassum, S.A., & Probert, S.D. (1987) Vertical-axis wind turbine: a modified design. Applied Energy, 28(1), 59-67.
2. Abraham, J.P., Mowry, G.S., Plourde, B.P., Sparrow, E.M., & Minkowycz, W.J. (2011). Numerical simulation of fluid flow around a vertical-axis turbine. Journal of Renewable and Sustainable Energy, 3(3), 1-13.
3. Kim, S.E., Choudhury, D., & Patel, B. (1997) Computations of Complex Turbulent Flows Using the Commercial Code FLUENT. In. Proceedings of the ICASE/LaRC/AFOSR Symposium on Modeling Complex Turbulent Flows, Hampton, Virginia.
4. Saha, U.K., Thotla, S., & Maity, D. (2008). Optimum design configuration of Savonius rotor through wind tunnel experiments. Journal of Wind Engineering and Industrial Aerodynamics, 96(8-9), 1359-1375.
5. Kamoji, M.A., Kedare, S.B., & Prabhu, S.V. (2009). Experimental investigations on single stage modified Savonius rotor. Applied Energy, 86(7-8), 1064-1073.