International Journal of Mechanical Engineering and Robotics Research

ISSN 2278 – 0149 www.ijmerr.com
Vol. 4, No. 1, January 2015

Research Paper

ANALYSIS OF BLADE AERODYNAMICS AND OBTAINING THE FORCES OF 1.5 KW HORIZONTAL AXIS DUAL ROTOR WIND TURBINE USING THE ANSYS FLUENT SOFTWARE

Hitendra Kurmi[1]*, Anurag Gour[1] and Mukesh Pandey[1]

*Corresponding Author: Hitendra Kurmi, ✉ htr.ptl@gmail.com

The exploitation of small horizontal axis wind turbines provides a clean and viable option for energy. Although great progress has been achieved in the wind energy sector, there is still potential space to reduce the cost and improve the performance of small wind turbines. An enhanced understanding of how small wind turbines interact with wind turns out to be essential. Currently large no research has concentrated on improving the aerodynamic performance of wind turbine blade through wind tunnel testing and theoretical studies. However wind turbine simulation through Computational Fluid Dynamics (CFD) software offers expensive solution to aerodynamics blade analysis problem.

Keywords: Wind energy, Aerodynamics of the wind turbine, Small horizontal axis wind turbine, Blade design and analysis, Ansys fluent, Computational Fluid Dynamics (CFD)

INTRODUCTION

Energy is important to human civilisation development. With progress of economics and socialism, there is an expanding demand on renewable energy to secure the supply of energy such as solar, wind power. Wind energy is a plentiful resource in comparison with other renewable resources. In wind energy wind power is converted to electrical energy and this machine is called the wind generator. Power in the wind comes from the transformation of the air that is driven by the heat of the sun, which is abundant, clean and renewable. As the end of 2011 the installed capacity for wind power worldwide (cumulative total) was 238 GW. Of this, India's installed capacity (16.1 GW) placed in fifth in the world and second in Asia after China (India's Potential Installed Capacity for Wind Power (Cumulative Total as the End of 2011)).

WIND TURBINE AERODYNAMICS AND BLADE DESIGN THEORY

A wind turbine consists of several main parts,

[1] School of Energy and Environmental Managment, UTD, RGPV, Bhopal, MP, India.

i.e., the rotor, generator, gearbox, yawing system, control system and so on. The rotor is driven by the wind and rotates at predefined wind speed, so that the generator can produce electric energy output under the regulation of the control system. To extract the maximum kinetic energy from wind, researcher put much effort on the design of effective blade geometry. This research aims to evaluate the aerodynamics performance of small horizontal-axis wind turbine blades through two and three dimensional Computational Fluid Dynamics (CFD) analysis (Ansys-Fluent (ICEM CFD 14.5)).

Blade Element Theory

Blade element theory which is published in 1948 given by Glauert. Blade is divided into several section sweeps an annular area when the turbine rotor rotates. These section are separated and no interaction between each other. By calculating the torque and thrust forces using wind tunnel tested airfoil lift and drag coefficient for each section, total power can be calculated by integral of infinite no. of elements (Schreck and Robinson, 2007).

Lift, Drag and Moment

Given an aerofoil, the design tip speed ratio is the first parameter that used in a blade design procedure, which is generally taken as 6-8 in modern wind turbines. As a higher lift coefficient means a larger lift force, a higher drag coefficient means a larger drag force, a turbine with the aerofoil of a higher lift coefficient and a lower drag coefficient is expected to produce more power with better load conditions. The maximum lift-to-drag ratio occurs should be considered to be the optimal attack angle. This optimal attack angle, which is equal to the angle of relative wind minus twist angle and pitch angle at all section when the blade geometry is optimal designed according to the BEM theory, should be used in the design to calculate ideal power coefficient.

The BEM theory divide a blade into several from root to tip and the total power coefficient is calculated by integrating the power coefficients at these sections, as described in

$$C_p = \left(8/\lambda^2\right)\int_{\lambda_h}^{\lambda} F\sin^2\phi\left(\cos\phi - \lambda_r\sin\phi\right)$$

$$\left(\sin\phi + \lambda_r\cos\phi\right)\lambda_r^{\,2}\left[1-\left(\frac{C_1}{C_d}\right)\cot\phi\right]d\lambda_r$$

Here, C_p is the power coefficient, C_1 is the lift coefficient, C_d is the drag coefficient, λ is the tip speed ratio, λ_h is the speed ratio at hub (root), λ_r is the local speed ratio at position r/R, ϕ is the angle of relative wind, and F is the tip loss factor (Manwell McGowan, 2002).

Betz Limit

Betz who was a German physicist in 1919, 0.593 is the maximum power efficiency of a wind turbine which converts the kinetic energy to mechanical energy, When the tip speed ratio reaches to 6, the efficiency is approximate 96%. It indicates that wind turbines with high tip speed ratio can extract more kinetic energy from wind by comparing with low tip speed ratio wind turbines.

Computational Fluid Dynamics (CFD) Based Approach for Horizontal Axis Wind Turbine

The success of any optimisation design is dependent on the clear definition of the design objective as well as the limitations on the solution space. Definition of the solution space is dependent on the extent of freedom of the design variables. Optimisation methodology is widely applied due to the rapid increase of multi-variable problems within engineering.

This method is extremely lengthy due to the iterative nature of CFD software and optimisation methods. Generally, the design geometry as well as the mesh of the flow field is optimised.This leads to a vast increase in design variables for optimisation. Furthermore, the automation of CFDwithin an optimisation program is an intricate task. In general, better CFD results are obtained when meshrefinements are tailored to the design geometry.

The Spalart-Allamaras model is a transport equation model for the eddy viscosity. The differential equation is derived by "using empiricism and arguments of dimensional analysis, Galilean invariance and selected dependence on the molecular viscosity". This model does not require finer grid resolution than the one required to capture the velocity field gradients with algebraic models (Transport Equation for the Spalart-Allamras Models Theory).

The transport equation for the working variable $\tilde{\nu}$

$$\frac{\partial \tilde{\nu}}{\partial t} + \tilde{u}\frac{\partial \tilde{\nu}}{\partial x_j} = Cb1\hat{S}\tilde{\nu}$$

$$+\frac{1}{\pm}\left\{\frac{\partial}{\partial x_j}\left(3 + \tilde{\nu}\right)\frac{\partial \tilde{\nu}}{\partial x_j} + Cb2\frac{\partial \tilde{\nu}}{\partial x_j}\frac{\partial \tilde{\nu}}{\partial x_j}\right\}$$

$$-Cw1f1\left(\frac{\tilde{\nu}}{d}\right)^2$$

METHOD

There are many commercial CFD softwares used in engineering, such as PHOENICS, STAR-CD, and ANSYS FLUENT/CFX and so on. Three main processor are the same which are Pre-Processor, solver and Post Processor. Setting of the governing equation is the precondition of CFD modelling, mass and momentum and energy conservation equation are three basic governing equations. After that, Boundary conditions are decided as different flow conditions and a mesh is created. The purpose of meshing model is discretised equation and boundary conditions into a single grid. The basic elements of two-dimensional unstructured grid. Finite Volume Method (FVM) is used in CFD software such as FLUENT and CFX. In this project used the software ANSYS FLUENT Non License Version Workbench 14.5.

In CFD software, wind turbines are simulated under the turbulent flows. The turbulence model contains one and two equations model. The one equation "Spalart-Allamaras" model and two equations "standard *k*-£" models are widely used in CFD softwares.

Table 1: CFD Model for Blade Airfoil

Airfoil	**Naca Airfoil 21% Thickness**
Simulation	Steady simulation
Fluid material	Air
Temperature	300 K
Kinematic viscosity	Sutherland
Density	1.25 kg/m^3
Pressure	101325 pa
Solver	Pressure-based
Turbulent model	Spalart-Allamaras
Interpolating scheme	Pressure (standard) Density (second order upwind) Modified turbulent viscosity (second order upwind)
Boundary condition	Pressure-far-field

Create the geometry in design modeller of the FLUENT or pre-processor of the CFD. In the geometry design the airfoil profile and create the domain to anysis of the wind behaviour. After the design modeller meshing is generated.

Figure 1: Meshing Geometry of the Front Rotor

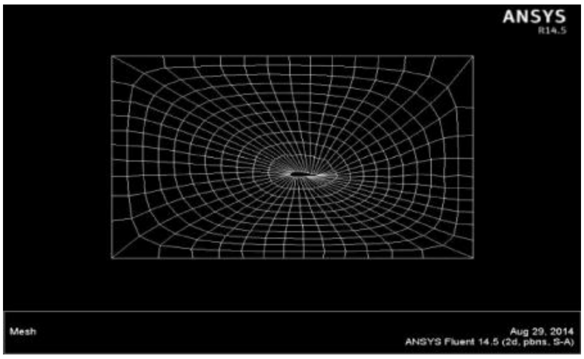


Figure 2: Meshing Geometry of the Rear Rotor

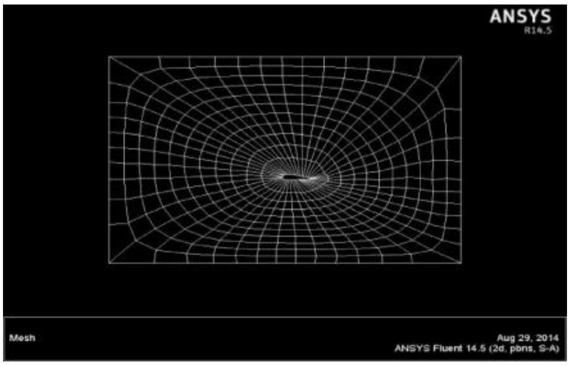


Figure 3: Pressure Contour at AOA 10° for Front Rotor

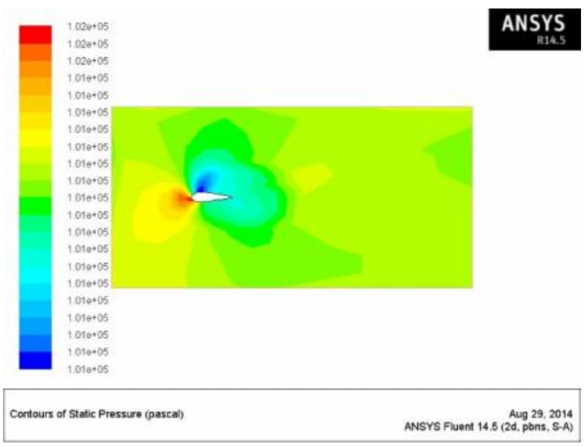
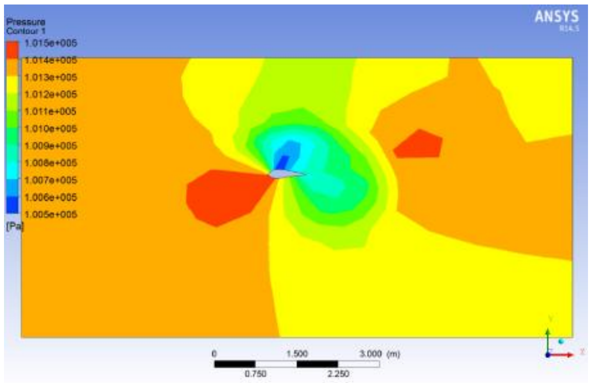


Figure 4: Pressure Contour at AOA 10° for Rear Rotor



In Ansys fluent, the pressure-based solver is used for the low speed incompressible flow. In the pressure-based solver, pressure and pressure corrections are used for the calculation of pressure field.

RESULTS AND DISCUSSION

The Design of the Dual Rotating Wind Turbine blades is analysed by using the Ansys Fluent (ICEM CFD 14.5) Software. The author has assumed the value of blade length 1.5 m length, front rotor chord length is 1.0 m and rear rotor chord length is 0.70 m and Rated wind speed of RGPV Bhopal is 10 m/s.

In this paper a dual rotor horizontal axis wind turbine blade analysed of the both front and rear rotor with Naca4421 air profile for 10° angle of attack and wind speed is 10 m/s is given by the Table 2.

In Table 2 show the output of the wind turbine generator is 1.5 kW at the wind turbine efficiency of 37%. Here take the designed

Table 2: Wind Turbine Data

S. No.	Basic Parameter	Unit	Value
1.	Wind turbine generator output	W	1500
2.	Design wind speed	m/s	10
3.	No. of blades		6
4.	Power coefficient of the wind turbine		0.37
5.	Air density	Kg/m^3	1.225

rated wind speed is 10 m/s, which is calculated by the wind anemometer located as the college campus.

The CFD analysis is carried out of the ANSYS FLUENT software. Figure of the pressure contour for both rotor at AOA 10° as shown by the figure and result obtain by the solver CFD. This is given the lift force, drag force and total forces of the blade surface which is given by the Table 3.

Table 3: Force Calculation

	Front Rotor	Rear Rotor
Total forces	103.89844 N	102.69141 N
Coefficient of pressure	169.6301	17.819525
Forces on airfoil top	7.18387 N	–1.8091 N
Forces on airfoil bottom	27.048 N	13.7922 N
Lift force	103.89844 N	103.91016 N
Drag force	31.608582 N	11.1931 N

CONCLUSION

A case study of a 1.5 kW horizontal axis wind turbine blade design has been carried out with an existing generator and a wind speed resource. With a rated wind speed of 10m/s and a generator of 1 kW and 120 RPM, a rotor of 1.6 m radius and the NACA airfoil has been applied. It has been predicted to be with a power coefficient of 0.37 at the tip speed ratio of 7 based on the BEM theory. A further structure analysis and testing will be developed in the future.

REFERENCES

1. Ansys-Fluent (ICEM CFD 14.5) Workbench, the Simulated Result Can be Imported into the Static Structural Analysis.
2. Fluent Inc., “Transport Equation for the Spalart-Allamras Models Theory”, FLUENT 6.3 User’s Guide FLUENT Inc.
3. Manwell J McGowan (2002), “Wind Energy Explained: Theory, Design and Application”, pp. 83-138, John Willey & Sons Inc.
4. Schreck S J and Robinson M C (2007), “Horizontal Axis Wind Turbine Blade Aerodynamics in Experiments and Modelling”, *IEEE Transactions on Energy Conversion*, Vol. 22, No. 1.