IJETIE
International Journal of Emerging Technology and Innovative Engineering
Volume I, Issue 3, March 2015
ISSN: 2394 - 6598
www.ijetie.org

AERODYNAMIC ANALYSIS OF BLADE 1.5 KW OF DUAL ROTOR HORIZONTAL AXIS WIND TURBINE

HITENDRA KURMI
Research scholar, School of Energy and Environmental Managment,UTD, RGPV
Bhopal,MP,INDIA
htr.ptl@gmail.com

ANURAG GOUR
Assistant Professor, School of Energy and Environmental Managment,UTD, RGPV
Bhopal,MP,INDIA
anurageem@gmail.com

MUKESH PANDEY
Professor, Dean &Head , School of Energy and Environmental Managment,UTD, RGPV
Bhopal,MP,INDIA
mukeshrgtu@yahoo.co.in

ABSTRACT
A wind turbine converts kinetic energy into mechanical power through a rotor, and then converts the mechanical power into electric power through a generator which is linked to the rotor with and without a gearbox. Various types of wind turbines are designed to take advantage of wind power based on the principle of aerodynamics. Depending on the wind turbine rotor orientation, there are two types of wind turbine , horizontal axis wind turbine(HAWT) and vertical axis wind turbine(VAWT).When considering installation sites, there are onshore(free standing or building mounted) and offshore wind turbines. Based on the operation scheme, wind turbine can be divided into stall regulated (fixed pitch) wind turbine and pitch controlled (variable pitch) wind turbines. According to the relative flow direction of the wind turbine rotor, horizontal axis wind turbine are either upwind or downwind turbines.

Keywords
Blade moment theory, Wind speed, Horizontal axis wind turbine, Ansys Fluent (ICEM- CFD), Wind Energy

1. INTRODUCTION
Energy is essential to human civilisation development. With progress of economics and socialisation, there as an expanding demand on renewable energy resources to secure energy supply, such as solar power ,wind power ,tide and wave power etc. As a clean renewable resources. Wind power plays a important role in modern life. According to the British wind energy Association (BWEA). It was estimated that wind power production met 12.2% of electricity demand in the UK around the end of 2011, and the government aims to reach a target of 20% from renewable in 2020[1] 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 one of the most popular renewable energy resources wind power exploitation is growing rapidly. At the beginning of 2006, the total installation of wind capacity reached 59206MW worldwide [2]. In june2011,a total installation of 5.560MW was operational in the UK and is predicted by RENEWABLEUK that in 2012 the annual wind power capacity will increase to 1.2GW[3].

1.1 The Role of Aerodynamics in Wind turbine Design

A wind turbine is a complex system which consists of several components, including a rotor, a transmission system, a generator, a nacelle, a tower and other electro-mechanical subsystems. The rotor blades are the most important components. In order to transfer wind energy into mechanical power, the blade is designed as aerodynamics geometry with nonlinear chord and twist angle distributions. The section view of a wind turbine blade is of an aerofoil shape, which is generated to high lift and low drag forces. The shape of the blade is vital as it determines the energy captured, and the loads experienced. The study of interaction between wind flows and wind turbines is wind turbine aerodynamics which plays an important role in wind turbine design and analysis.

The most popular theory in wind turbine aerodynamics is the Blade Element Moment (BEM) Theory which was firstly published by Glauert in 1948[3]. In the blade is divided into several section sweeps an annular area when the rotor rotates. These annuli 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 annulus, the total power and thrust forces can be obtained by integral of an infinite number of sections/elements. This is a great development in the history of the wind turbine aerodynamics, which relates the blade geometry to power and thrust forces using lift and drag coefficients. It provides a principle to design optimal blade geometry.

IJETIE
International Journal of Emerging Technology and Innovative Engineering
Volume I, Issue 3, March 2015
ISSN: 2394 - 6598
www.ijetie.org

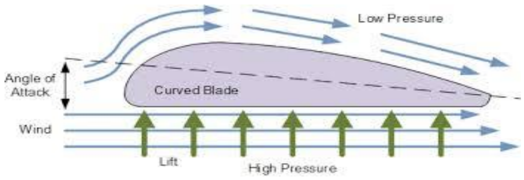

Fig 1: Air flow diagram

Lifting line and Computational Fluid Dynamics (CFD) methods are widely applied in airfoil aerodynamic analysis. All these numerical methods are employed in near wake and far wake analysis.

1.2 Power of the wind

$P_{wind} = {}^{1}/_{2}\, \rho A V^3$

Where ρ =1.225 Kg/m^3 is density of the air

A rotor swept area

V is the rated wind speed

1.3 Rotor design

In the design procedure, the main parameters such as aerofoil type, design tip speed ratio, rated wind speed, rotor diameter, should be consider first before conducting blade geometry optimization.

1.3.1Aerofoil Blade selection

Aerofoil for HAWT is often designed to be used at low attack angle, where the drag coefficient is much lower than the lift coefficient.

For a stall regulated wind turbine, it is better to choose an aerofoil shape to make that stall occurs gently after the maximum lift-to-drag point. Design a wind turbine for a specific site should not only include an optimum geometry with the maximum power but also the detailed power coefficient curve which is a function of wind speeds or the tip speed ratio. With more accurate aerodynamic coefficients at high attack angles, the more accurate design and performance prediction can be obtained. But the aerodynamic coefficient of a rotating aerofoil is different from the ones of a linear moving aerofoil. The coefficients from wind tunnel testing are acceptably accurate in steady flow, but in stall condition, these coefficients are lack of accuracy or there is no coefficient measured at very high attack angles at all .The low maximum lift coefficient, 21% thickness to chord ratio NACA0021 aerofoil has been extensively used in HAWT.

1.3.2 Design tip speed ratio

The tip speed ratio is the ratio of the blade tip speed over wind speed. It is a significant parameter for wind turbine.

Tip speed ratio $\lambda = \omega R / v_0$

ω is the angular velocity of the wind turbine rotor, R is radius of the rotor and v_0 is the wind speed

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.[4]

Blade Element Theory

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

$C_p = ({}^{8}/_{\lambda^2})\int_{\lambda_h}^{\lambda} F\sin^2\varphi(\cos\varphi - \lambda_r\sin\varphi)(\sin\varphi + \lambda_r\cos\varphi)\lambda_r^2 [1-(\frac{C_l}{C_d})\cot\varphi]\, d\lambda_r$[5] Here, C_p is the power coefficient, C_l 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.

From the above equation, it can be seen that there is a relationship between the ideal total power coefficient and different tip speed ratios.

1.3.3 Rated wind speed

The rated wind speed is the wind speed at which the wind turbine is generating its rated power. And the wind power is proportional to the cube of the wind speed; high wind speed means high power can be produced. However, a high rated wind speed is not always a good choice as the annual power output is also a function of the local wind speed distribution.

1.3.4 Rotor size

Given a rated wind speed, Vr, in m/s, the power extracted by the rotor from the wind, is defined as:

$P = {}^{1}/_{2}\, C_p\, \rho\pi R^2 V^3$

Here C_p is the power coefficient which is 0.37. ρ is the density of air which is 1.225 kg/m^3, R is the rotor radius.

METHODOLOGY

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 equation. 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[6].

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-ε” models are widely used in CFD softwares. 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[7].

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.

Table 1: Geometry of the Front rotor airfoil

Front Rotor radius	1.6m
Front Blade radius	1.44m
Chord length at root section	.70m
Chord thickness	21% of the radius at 30-40% position
Chord length at tip	.21m

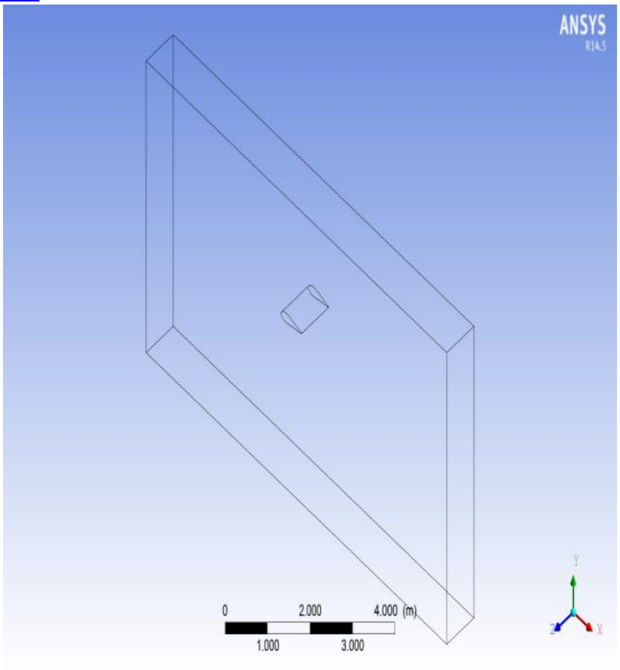

Fig 2: Geometry of the blade

Meshing of the geometry after creating the geometry, start the meshing modeller Meshing, sizing, and name selection are all of the process are held on this meshing modeler.

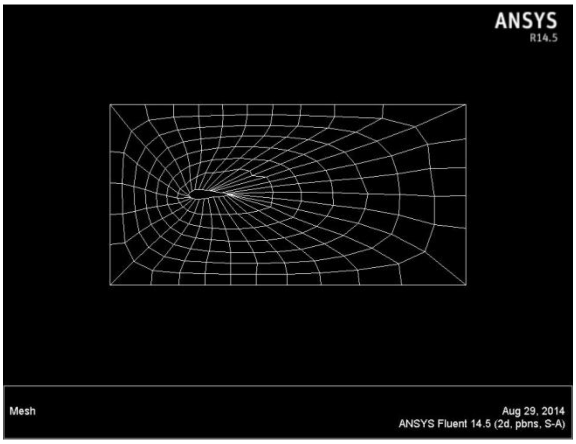

Fig 3: meshing of the geometry of front rotor blade

Table 2: Computational feature of the front rotor airfoil on CFD

Airfoil	NACA0021
Simulation	Steady simulation
Fluid material	Air
Temperature	300K
Kinematic viscosity	Sutherland
Density	1.25 kg/m^3

International Journal of Emerging Technology and Innovative Engineering
Volume I, Issue 3, March 2015
ISSN: 2394 - 6598
www.ijetie.org

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

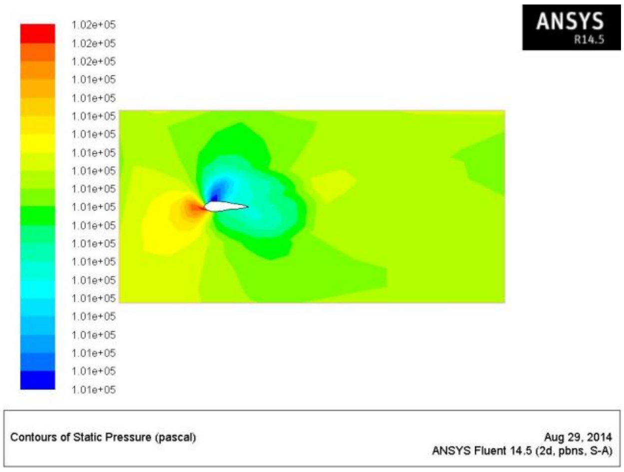

Fig 4: Pressure contour at AOA is 10 degree

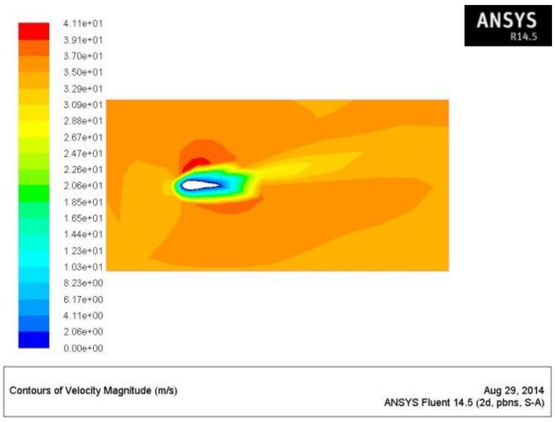

Fig 5: velocity contour at angle of attack 10 degree

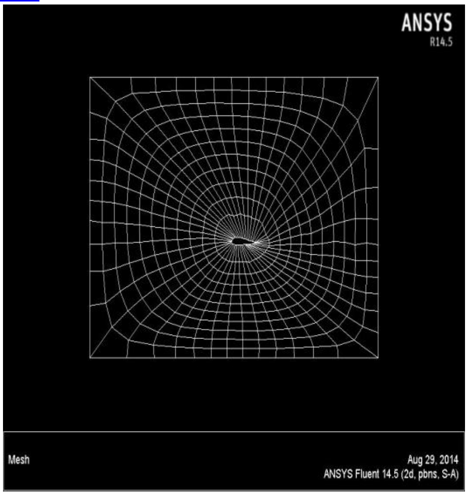

Fig 6: Meshing of the rear rotor.

Table 3: Geometry of the rear rotor airfoil

Rear rotor radius	0.82m
Chord length at rear blade at root section	0.4m
Chord thickness of the blade	21% of the chord length

Table 4: computational features of the rear airfoil on CFD

Airfoil	NACA0021
Simulation	Steady simulation
Fluid material	Air
Temperature	300k
Kinematic viscosity	Stherland
Density	1.25kg/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

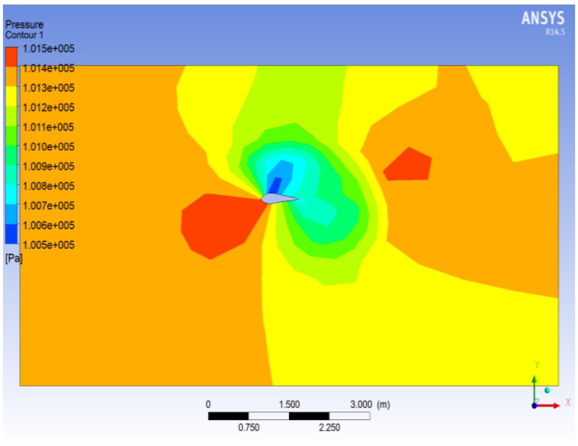

Fig 7: pressure contour at CFD post

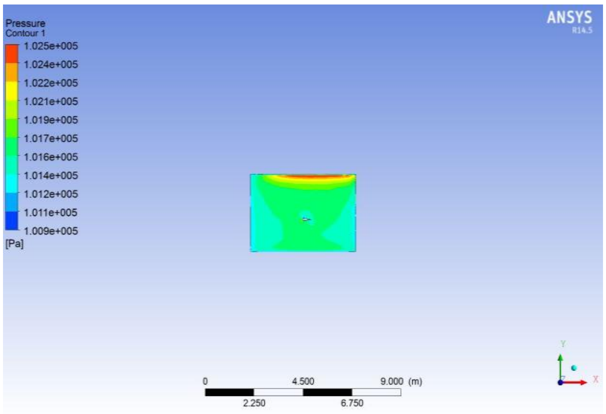

Fig 8: pressure contour in the rear rotor

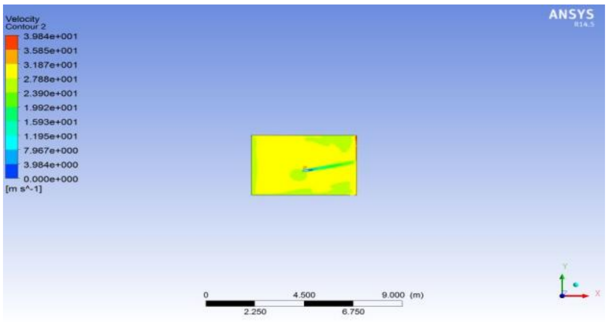

Fig 9: Velocity contour at 10 degree angle of attack

Result 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 1.0m and rear rotor chord length is .70 m and Rated wind speed of RGPV Bhopal is 10m/s.

Table 5: Two-dimensional aerofoil modelling, the simulated result

Sr.no.	Basic parameter	Unit	Value
1	Wind turbine generator output	W	1500
2	Design wind speed	m/s	10
3	No. of blades per rotor		3
4	Power coefficient of the wind turbine		.37
5	Air density	Kg/m^3	1.225

Conclusion

A project 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 7m/s and a generator of 1.kW and 120 RPM , a rotor of 1.6m 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.

Acknowledgement

This work is outcome of the Research project- “Enhanced Power Performance of HAWT using Optimum Dual Rotor Wind Turbine Design” at Rajiv Gandhi Technological University of India.MP. INDIA funded and sanctioned by All India Council Of Technical Education, Government Of India. The work is done under the Guidance of Prof. (Dr.) Mukesh Pandey, Dean RGTU and Anurag Gour Asst. Prof.RGTU.

6, References

[1] RenewableUK 2011, “state of the industry report”, http://www.bwea.com/pdf/publications/SOI_2011.pdf.Accessed on August1, 2012.

[2] RenewableUK 2012, “Wind farms hit high of more than 12% of UK electricity demand”, http://www.bwea.com/media/news/articles/pr201206.html. Accessedon August 1.2012

[3] A., “Optimal Design of Horizontal-Axis Wind Turbines Using Blade-Element Theory and Evolutionary Computation,” Journal of Solar Energy Engineering, Vol. 124, Nov 2002, pp. 357-363 Bermúdez, L., Velázquez, A., Matesanz,

[4] N.S.cetin, M.A.Yurdusev,R.Ata, and A.Ozdemir, “Assessment of optimum tip speed ratio of wind turbines,”

Mathematical and Computational Applications, vol. 10, no. 1.

[5] Schreck .S.J & Robinson M.C. 2007, “Horizontal axis wind turbine blade aerodynamics in experiments and modelling”, IEEE transactions on energy conversion vol.22, no.1.

[6] Ansys-Fluent (ICEM CFD 14.5) workbench, the simulated result can be imported into the static structural analysis.

[7] Carlo ennico carangid, 2008, ‘Wind turbine functioning and aerodynamics’, CFD-RANS study of Horizontal Axis Wind Turbine, Italy.