INTERNATIONAL JOURNAL OF
ENERGY AND ENVIRONMENT

Volume 4, Issue 5, 2013 pp.825-834
Journal homepage: www.IJEE.IEEFoundation.org

CFD analysis of horizontal axis wind turbine blade for optimum value of power

Monir Chandrala, Abhishek Choubey, Bharat Gupta

Department of Mechanical Engineering, Oriental Institute of Science & Technology, Bhopal (MP), India.

Abstract
With the shortage of fossil fuels, alternative energy has been thrust into the national spotlight as a major necessity in order to keep up with the increasing energy demands of the world. Wind energy has been proven one of the most viable sources of renewable energy. A wind turbine is a rotary device that extracts energy from the wind. Rotor blade is a key element in a wind turbine generator system to convert wind energy into mechanical energy. In this paper rotor blade is made up of single airfoil NACA 0018. The CFD analysis of NACA 0018 airfoil is carried out at various blade angles at 32 m/s wind speed. The analysis showed that blade angle 10º gives optimum power. The pressure and velocity distributions are plotted. These results are compared with wind tunnel experiment values.

Keywords: NACA airfoil; CFD; Wind tunnel test.

1. Introduction

A wind is one of the renewable energy sources is a free, clean and inexhaustible energy source. A wind turbine is a rotary device that extracts energy from the wind [1]. Wind energy has been shown to be one of the most viable sources of renewable energy. With current technology, the low cost of wind energy is competitive with more conventional sources of energy such as coal [2]. Rotor blade is a key element in a wind turbine generator system to convert wind energy in to mechanical energy [3]. Wind machine operates in an environment totally different than airplane wings characterized with continually changing wind speed and direction. Since the power contained in a moving air stream is proportional to the square of the rotor diameter and to the cube of the wind speed, the rotor blades must be carefully designed in order to optimally extract this power and convert it into torque that drives the electrical generator.

2. Airfoil nomenclature

Figure 1 shows airfoil nomenclature and its terminology is as following:
Chord length – length from the LE to the TE of a wing cross section that is parallel to the vertical axis of symmetry
Mean camber line – line halfway between the upper and lower surfaces
Leading edge (LE) is the front most point on the mean camber line,
Trailing edge (TE) is the most rearward point on mean camber line
Camber – maximum distance between the mean camber line and the chord line, measured perpendicular to the chord line
- 0 camber or uncambered means the airfoil is symmetric above and below the chord line

Thickness – distance between upper surface and lower surface measured perpendicular to the mean camber line [4].

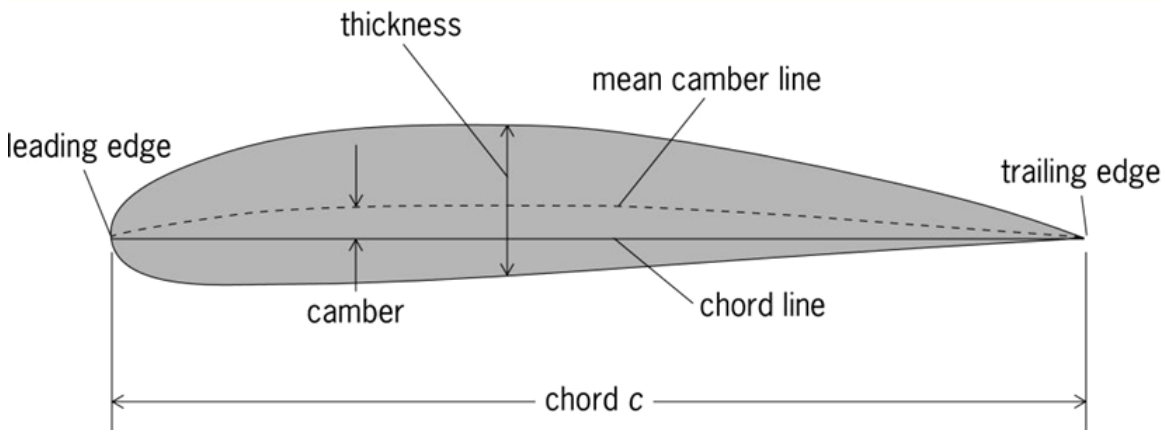

Figure 1. Airfoil nomenclature

3. NACA 0018 airfoil profile

In this paper, NACA 0018 airfoil is used. The blade is made up of single airfoil. The chord length varies from throughout of its length. For the analysis airfoil chord length 160 mm and span 290 mm are considered.

3.1 NACA four digit series (0018)

- First number is camber in percentage of chord (0)
- Second number is location of maximum camber in tenths of chord measured from LE (0)
- Last two digits give maximum thickness in percentage of chord (18) [5].

4. CFD analysis procedure

Design foil software is used to obtain airfoil coordinate [6]. Table 1 shows NACA 0018 airfoil coordinates for modeling of blade. Modeling of blade is carried out by using Solid works software, the data mentioned above of NACA airfoil profile is been used for this. Figure 2 shows modeling of HAWT blade. This file is saved as .IGES format. Now import this model IGES file into ANSYS CFX 12. Then after generate geometry in ANSYS CFX. Meshing the geometry is using hex dominant method. The element size is taken as 0.01 m. Figure 3 shows meshing of cavity domain for NACA 0018 airfoil.

Table 1. NACA 0018 airfoil coordinates for 160 mm chord length

X	Y	Z	X	Y	Z	X	Y	Z
160.00	0.00	0.00	29.36	13.52	0.00	29.36	-13.52	0.00
153.39	1.66	0.00	19.33	11.99	0.00	41.02	-14.29	0.00
142.08	3.83	0.00	11.17	9.79	0.00	54.06	-14.33	0.00
128.52	6.21	0.00	5.09	6.99	0.00	68.19	-13.67	0.00
113.73	8.54	0.00	1.30	3.71	0.00	83.09	-12.40	0.00
98.42	10.65	0.00	0.00	0.00	0.00	98.42	-10.65	0.00
83.09	12.40	0.00	1.30	-3.71	0.00	113.73	-8.54	0.00
68.19	13.67	0.00	5.09	-6.99	0.00	128.52	-6.21	0.00
54.06	14.33	0.00	11.17	-9.79	0.00	142.08	-3.83	0.00
41.02	14.29	0.00	19.33	-11.99	0.00	153.39	-1.66	0.00
						160.00	0.00	0.00

In the pre-processing fluid domain is selected. For the fluid air ideal gas is used. Here, domain is considered as stationary and heat transfer model is taken as total energy. K-ε turbulence model is used for 3-D as well as 2-D simulations of airfoil segment in Ansys cfx fluid flow module to predict the flow regime. The K-ε model is one of the most common turbulence models. The first transported variable is

turbulent kinetic energy; k. the second transported variable in this case is the turbulent dissipation factor, ε. The latter is a variable that determines the scale of the turbulence, whereas the first variable, k, determines the energy in the turbulence.

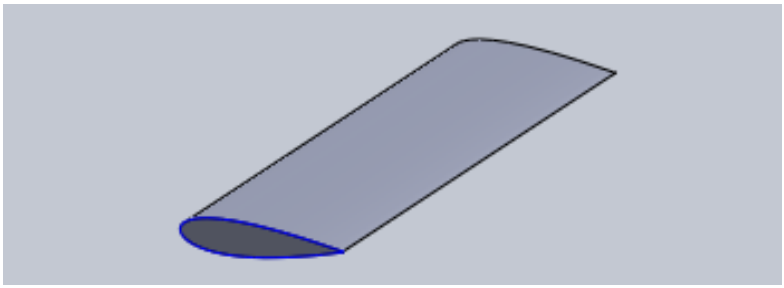

Figure 2. Modeling of HAWT blade

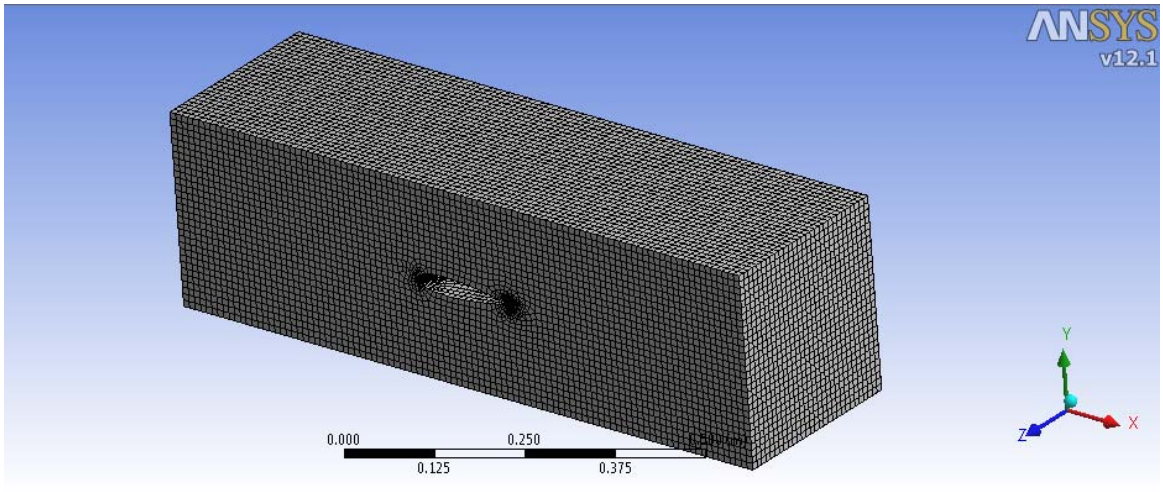

Figure 3. Meshing of NACA 0018 airfoil cavity domain

Define the boundary condition for the geometry like inlet boundary, outlet boundary, other wall are taken as symmetry boundary. Inlet velocity (v_u) as 32 m/s, (v_v) and (v_w) are taken as zero. Outlet pressure is atmospheric pressure. In the post processing of geometry gives different results like force, temperature, pressure, velocity, torque, Mach number etc. Here, pressure and velocity contours are shown in Figures 4-11. A simple solver was utilized. Calculation was done for the "linear " region [7-10].
From the Figures 4, 6, 8, 10 velocities are obtained for different blade angles. Power produced from the wind is given by following equation.

$$P_W = \frac{1}{2} \times \rho \times A \times v^3 \tag{1}$$

where, P_W is power produced from the wind, air density ρ is taken as 1.225 kg/m^3, area of blade A is taken as 0.160 m × 0.290 m, wind velocity v is taken as 32 m/s.
Table 2 is obtained from the calculation of power with respect to velocity.
From Table 2 graph of blade angle v/s power is obtained which is shown in Figure 12.

Table 2. Power at different blade angles

Sr. No.	Blade Angle	Velocity (m/s)	Density (Kg/m3)	Power (watt)
1	0	31.94	1.225	927.78
2	10	34.75	1.225	1192.60
3	15	31.65	1.225	901
4	30	33.19	1.225	1039

0 º angle

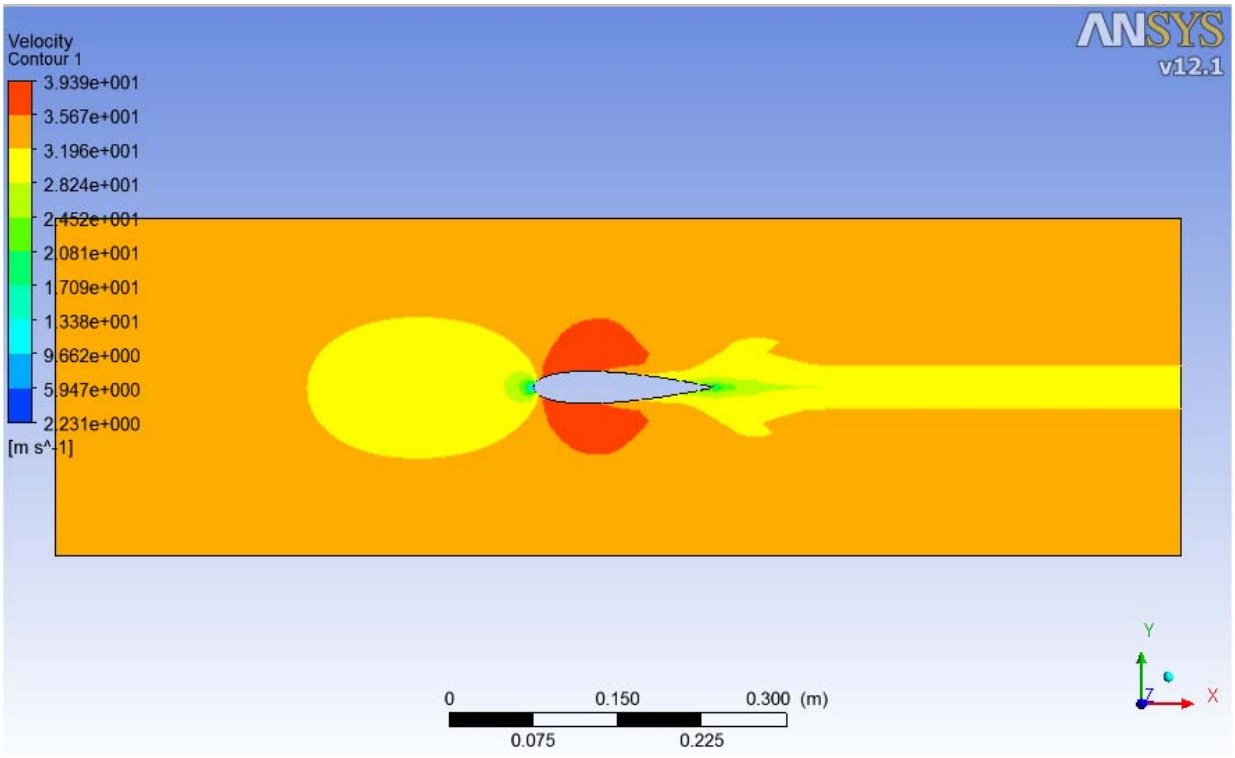

Figure 4. Velocity contour for 0 º blade angle

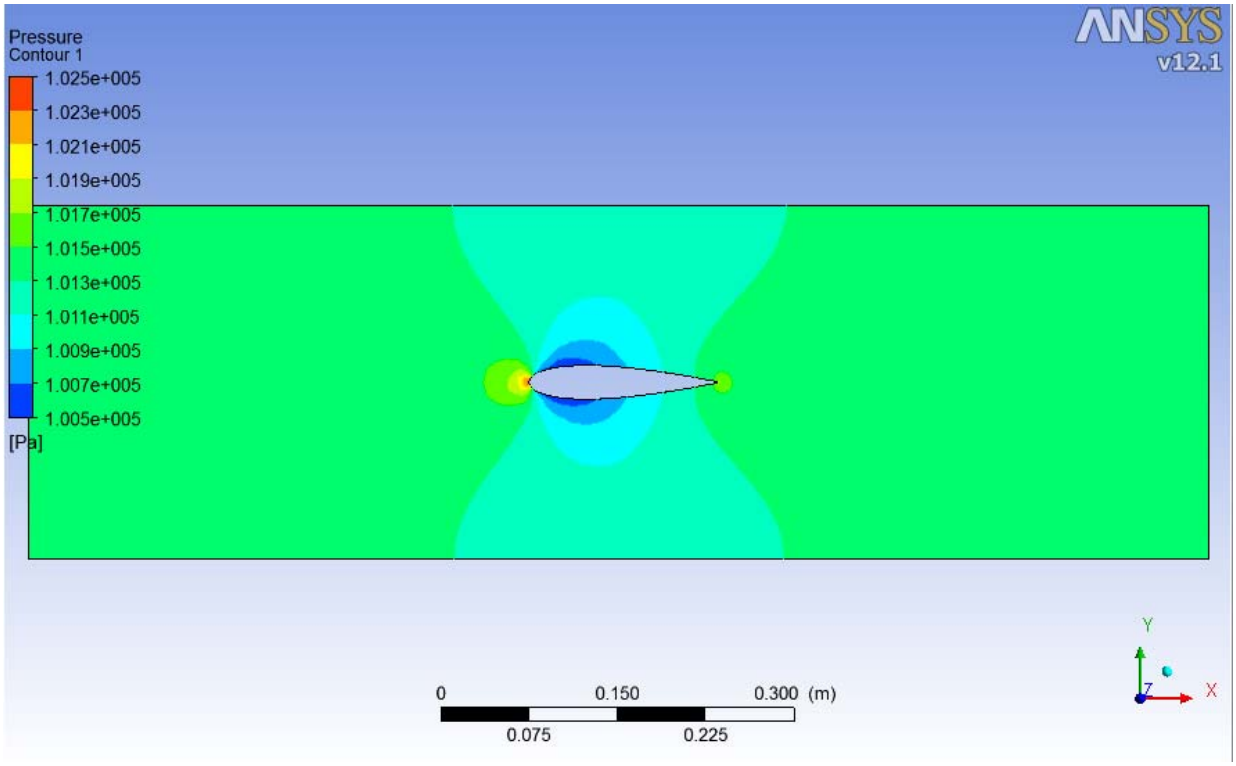

Figure 5. Pressure contour for 0 º blade angle

10 º blade angle

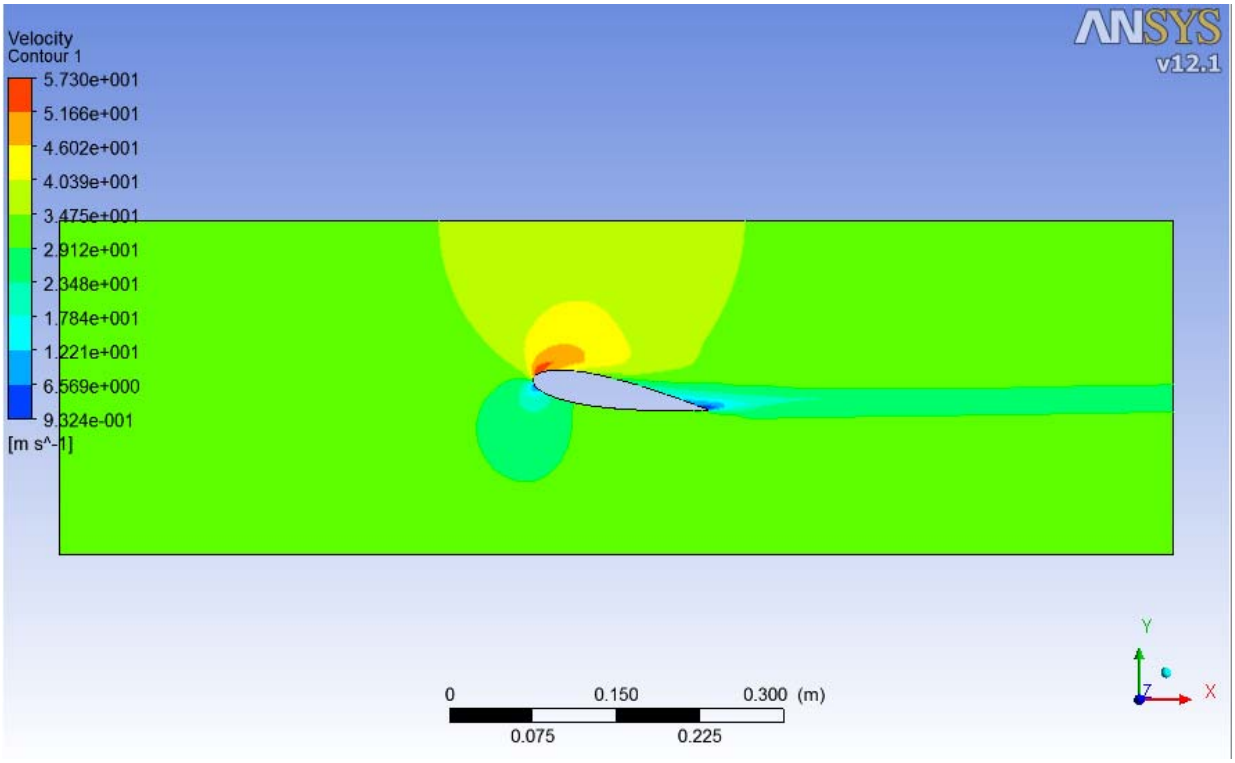

Figure 6. Velocity contoure for 10 º blade angle

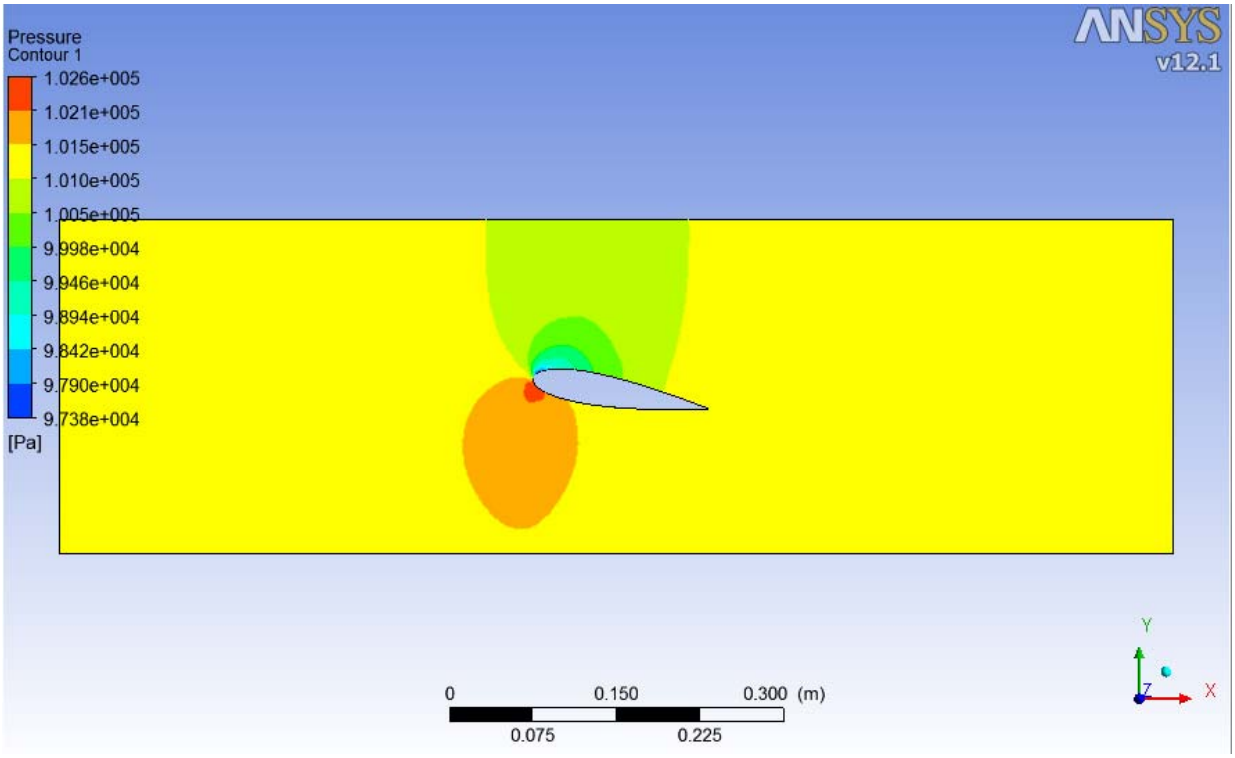

Figure 7. Pressure contour for 10 º blade angle

15 º blade angle

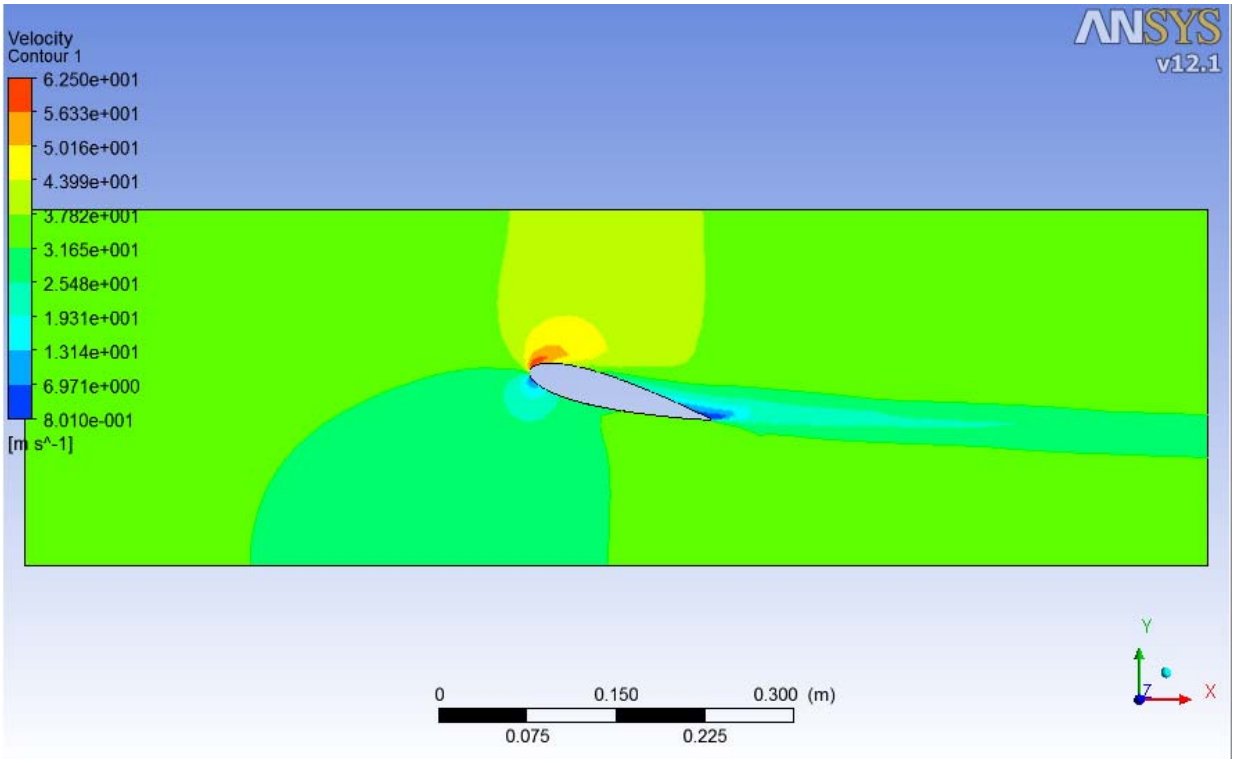

Figure 8. Velocity contour for 15 º blade angle

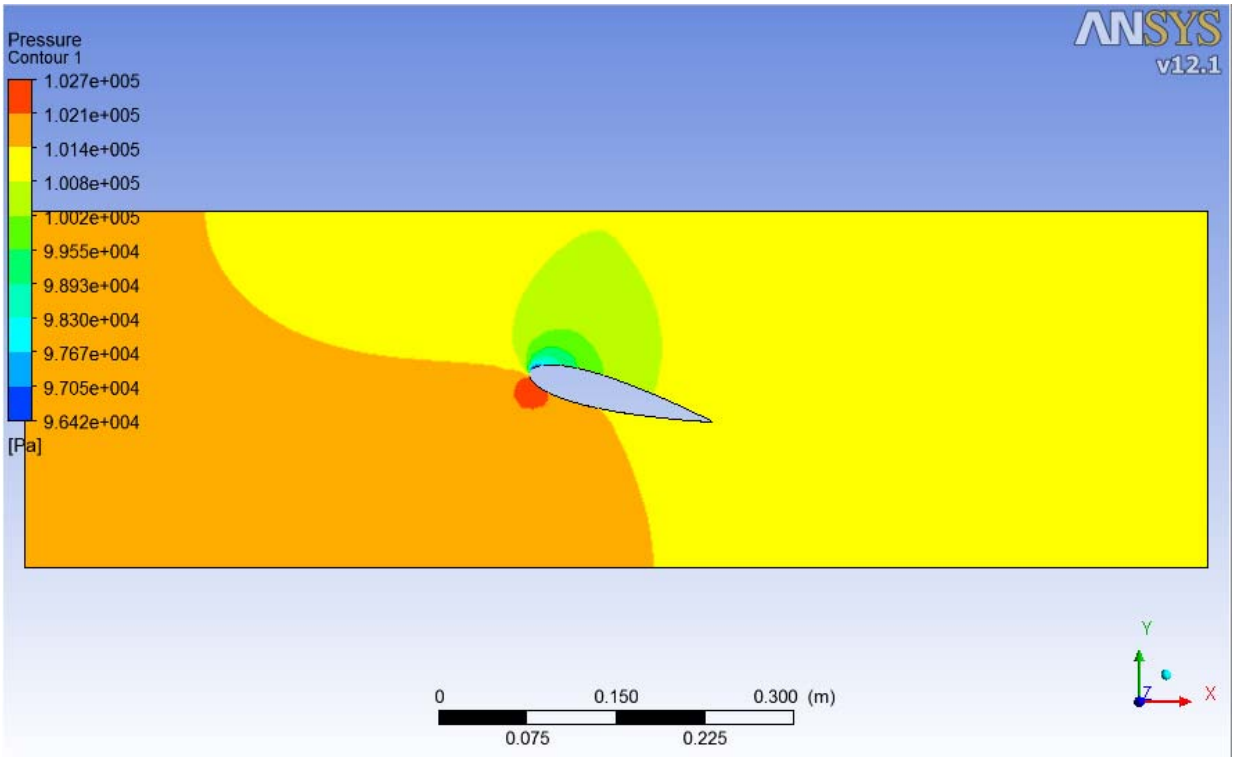

Figure 9. Pressure contour for 15 º blade angle

30 º blade angle

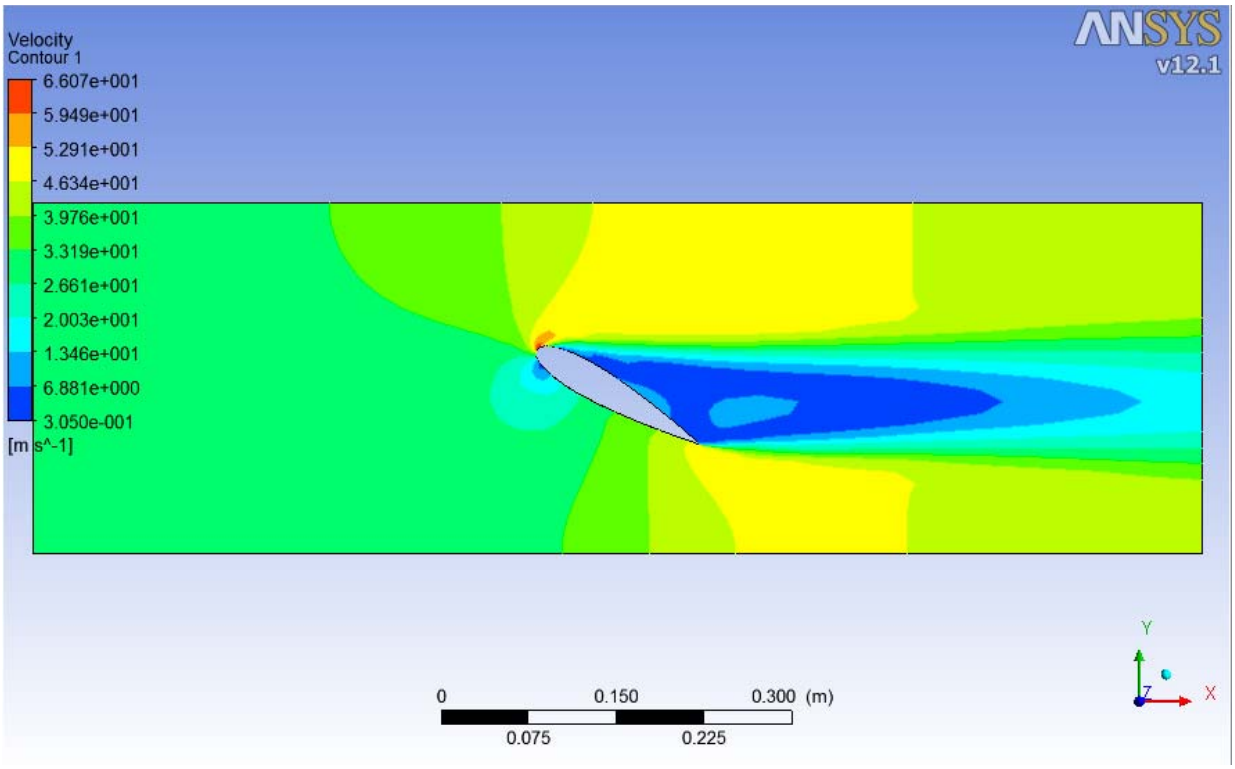

Figure 10. Velocity contour for 30 º blade angle

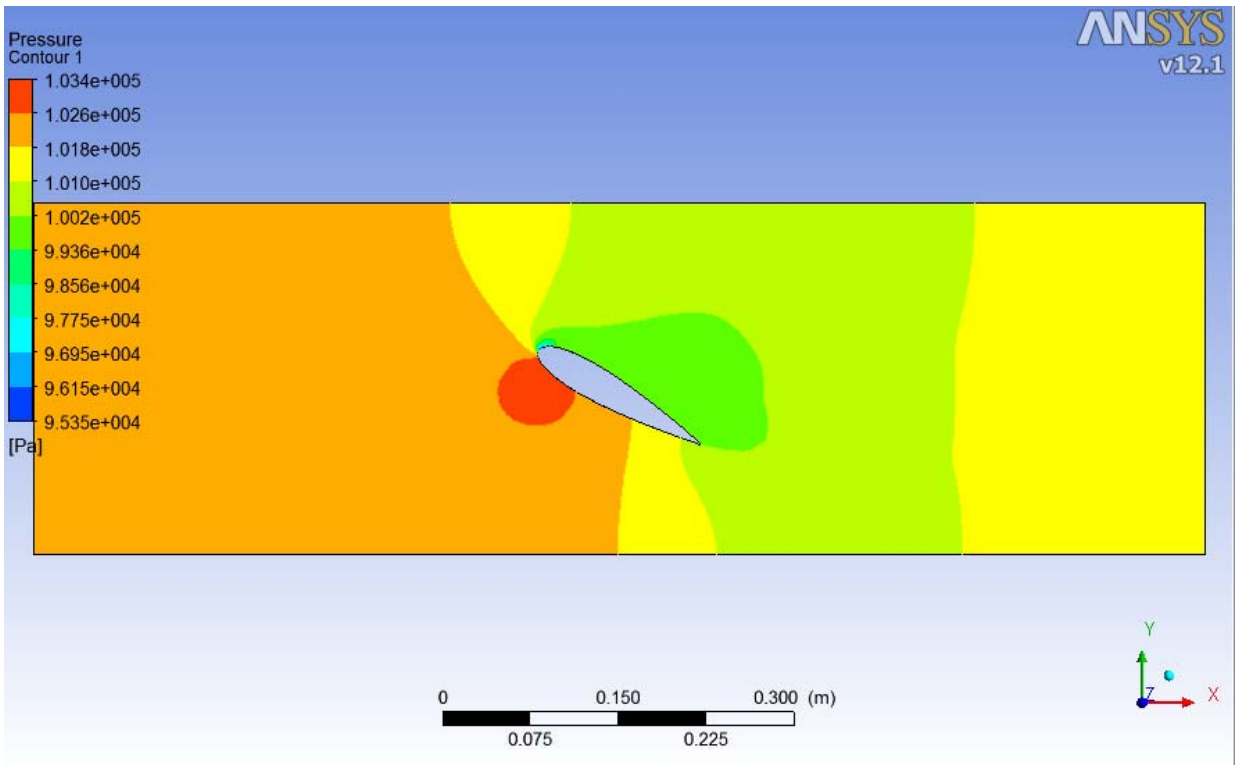

Figure 11. Pressure contour for 30 º blade angle

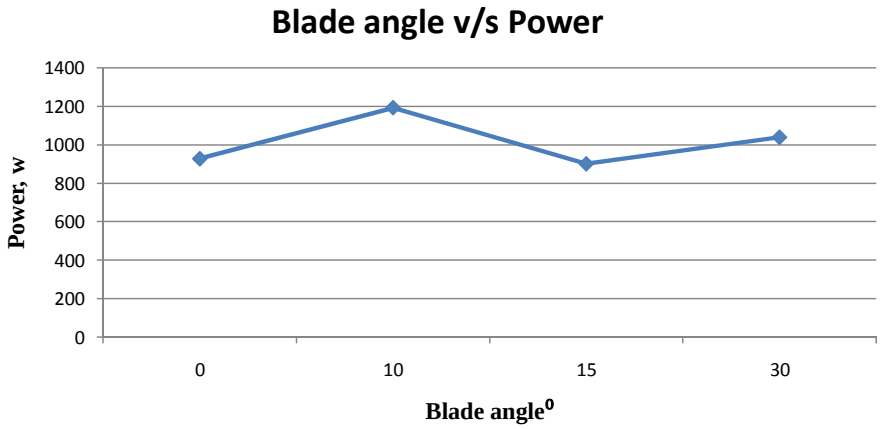

Figure 12. Blade angle v/s power

5. Wind tunnel testing

To compare the CFD analysis results with experiments values, wind tunnel test had been done. Here NACA 0018 airfoil (chord length 160 mm and span 290 mm) is selected for wind tunnel testing. The experiment setup is shown in Figure 13.

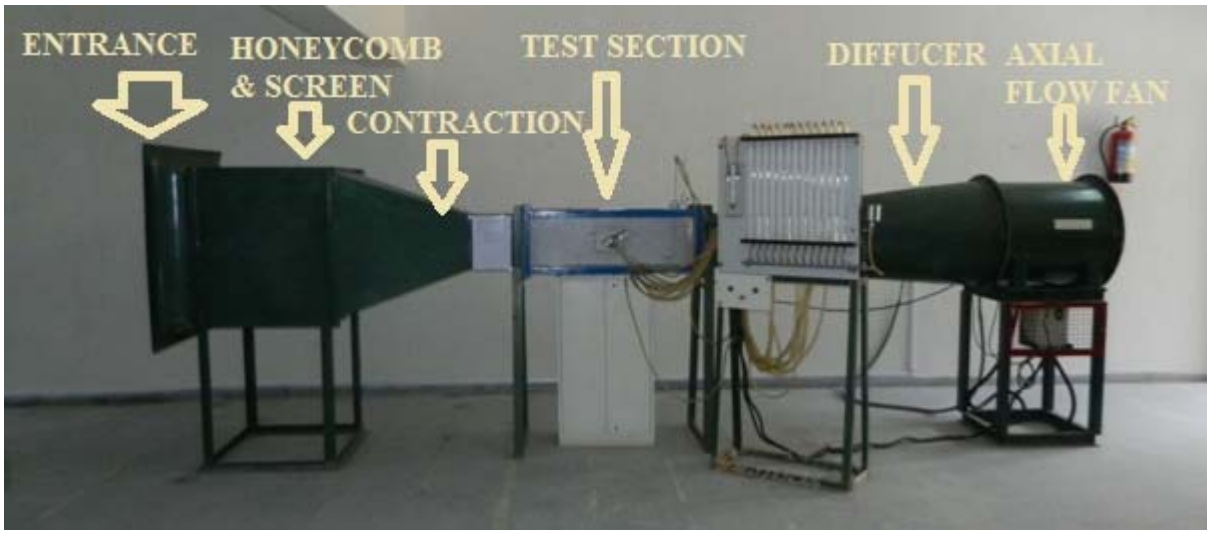

Figure 13. Wind tunnel testing equipment

5.1 Operating procedure

The following sections describe the initial steps to be followed in preparing the instruments.

Step-1: Divide the airfoil into 12 equal parts. Six parts are in upper surface and six parts are in lower surface. Connect manometer limbs to the various static pressure taps of the airfoil and fill the manometer reservoir with water.

Step-2: Initially, the lift and drag readouts should be set to zero with the model attached to it.

Step-3: The incidence angle is changed by loosening the bolts and manually positioning the airfoil at the required incidence angle.

To calculate coefficient of pressure (C_P) for the CFD analysis following equation is used.

$$C_p = \frac{\text{pressure at a point - atmospneric pressure}}{0.5 \times \rho \times v^2} \tag{2}$$

Calculated values of C_P for different blade angles are shown in Table 3, and Figure 14 shows it in chart form.

Table 3. CFD analysis C_p of NACA 0018 airfoil

Pressure points on airfoil	Blade angle 0 º	Blade angle 10 º	Blade angle 15 º	Blade angle 30 º
1	0.51	0.02	-0.36	-2.53
2	-0.37	-0.45	-0.78	-2.05
3	-0.72	-1.25	-1.35	-2.10
4	-1.07	-2.21	-2.26	-2.14
5	-1.30	-3.29	-3.37	-2.22
6	-1.12	-4.75	-4.81	-3.08
7	1.92	-1.67	-1.57	-5.10
8	-1.12	1.36	1.28	2.99
9	-1.25	0.47	0.66	2.18
10	-1.06	0.08	0.30	1.37
11	-0.70	0.01	0.07	0.82
12	-0.36	-0.01	-0.06	0.29

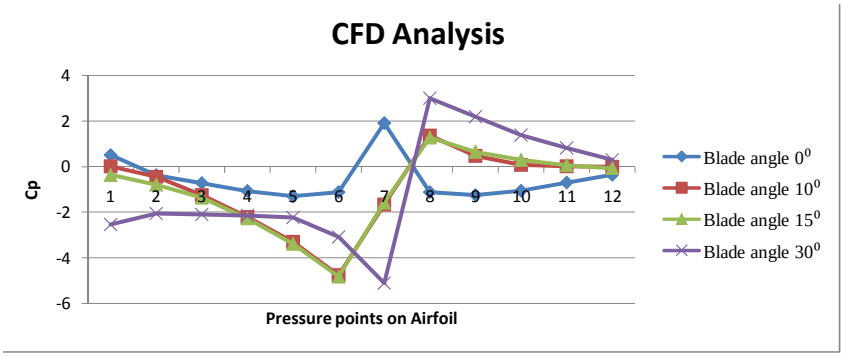

Figure 14. Cp of CFD analyses at different blade angle

5.2 Wind tunnel test Cp of NACA 0018 airfoil

Values of C_P for wind tunnel test for different blade angle is given in Table 4 and its graphical representation is shown in Figure 15.

Table 4. Wind tunnels test Cp of NACA 0018 airfoil

Pressure points on airfoil	Blade angle 0 º	Blade angle 10 º	Blade angle 15 º	Blade angle 30 º
1	-1.30	-1.30	-0.97	-2.47
2	-1.10	-1.47	-1.05	-2.22
3	-1.13	-1.38	-1.38	-2.27
4	-1.18	-2.22	-2.22	-2.13
5	-1.33	-3.30	-2.88	-2.30
6	-1.17	-3.80	-3.47	-3.05
7	-1.30	-1.80	-1.63	-3.30
8	-1.22	-1.43	-1.35	-1.13
9	-1.30	-1.30	-1.27	-1.22
10	-1.08	-1.80	-1.23	-1.33
11	-1.05	-1.72	-1.15	-1.47
12	-0.97	-1.63	-0.97	-1.55

6. Conclusion and future work

In this paper NACA 0018 airfoil is designed and analyzed for different blade angle at constant wind speed 32 m/s. The CFD analysis is carried out using ANSYS CFX software. The velocity and pressure distribution at various blade angles is shown in Figures 4-11. These results match with the wind tunnel experimental values. Hence the results are validated with the experimental work. The optimum value of power has been achieved at a blade angle 10 º for 32 m/s wind speed. In this paper flat blade with single airfoil is considered, analysis is also carried out using twisted blade with different airfoil with different wind speed.

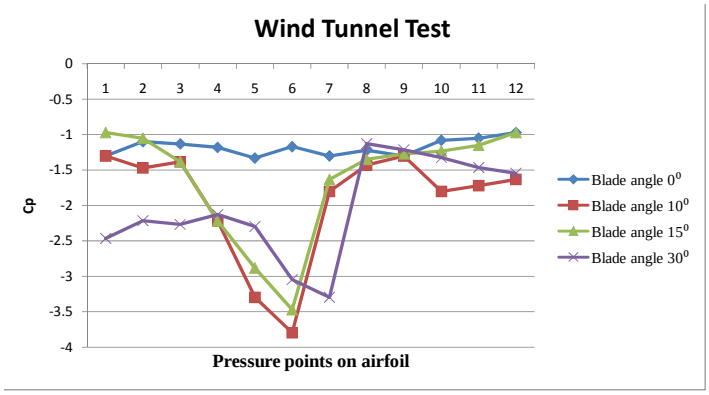

Figure 15. C_p of Wind tunnel test at different blade angles

References

[1] Wikipedia, the free encyclopedia, wind_ turbine.
[2] R.S. Amano, R.J.Malloy, "CFD analysis on aerodynamic design optimization of wind turbine rotor blade".
[3] Y.U. Sohn, Ch.H. Chun, Y.C. Kim, K.S. Han, "blade design of 750 kw direct drive wind turbine generator system".
[4] Airfoil_nomenclature.pdf
[5] NACA airfoil series.pdf
[6] S. Rajakumar, D. Ravindran ,"Computational Fluid Dynamics of Wind turbine Blade at Various Angles Of Attack and Low Reynolds Number".
[7] Thumthae C, Chitsomboon T. Numerical simulation of flow over twisted-blade, horizontal axis wind turbine. The 20th conference ofmechanical engineering network of Thailand.
[8] Ansys cfx tutorial 12.pdf
[9] Laursen J, Enevoldsen P, Hjort S. 3D CFD rotor computations of a multimegawatt HAWT rotor. European wind energy conference, Milan, Italy.
[10] Thumthae C, Chitsomboon T. "CFD Simulation of horizontal axis wind turbine in steady state condition". The 2nd Thailand national energy conference, Thailand.

Monir Chandrala is currently pursuing M.Tech in the Department of Mechanical Engineering at Rajiv Gandhi Proudyogiki Vishwavidyalaya University in Bhopal (M.P). He received B.E. in Mechanical Engineering from Saurashtra University in Gujarat in 2009. Monir's study focuses on wind energy and aerodynamics. He is working on CFD analysis of horizontal axis wind turbine blade for optimum values of power.
E-mail address: chandralamonir@gmail.com

Abhishek Choubey is currently working in Oriental Institute of Science & Technology, Bhopal, India as an Assistant Professor in Department of Mechanical Engineering. He received B.E in Mechanical Engineering from Rajiv Gandhi Proudyogiki Vishwavidyalaya University in Bhopal (M.P) in 2006 and M.Tech in Thermal Engineering from Rajiv Gandhi Proudyogiki Vishwavidyalaya. Bhopal (M.P). He mainly works on the wind energy and its utilization. Currently he focused on the FMEA analysis of horizontal axis wind turbine. Prof Choubey is the member of international association of engineers & African wind energy association.
E-mail address: Abhishekchby2@gmail.com

Bharat Gupta is currently working in Oriental Institute of Science & Technology,Bhopal,India as an Assistant professor in Department of Mechanical Engineering. He received M Tech in 1999 from Maulana Azad College of Technology Bhopal,(M.P), India. He works on the field of Mechanism of machine and non conventional energy resources. Prof. Gupta doing the PhD from MANIT, Bhopal, India.
E-mail address: Bharatgupta@oriental.ac.in

