Velocity Effect On Wind Turbine Blade Using Cfd

[1]Ch.Indira Priyadarsini, [2]V.Lakshmi Shilpa, [3]A.Akhil
[1]Faculty, Mechanical Engg.Dept., Chaitanya Bharathi Institute of Technology, Hyderabad
[2]M.E Student, Mechanical Engg.Dept., Chaitanya Bharathi Institute of Technology, Hyderabad
[3]M.E Student, Mechanical Engg.Dept., Chaitanya Bharathi Institute of Technology, Hyderabad

Abstract : Wind turbine blades are subjected to various aerodynamic loads; at present much research has focused on improving the aerodynamic performance of wind turbine blade through wind tunnel testing and analytical studies. These are much time consuming andrequire expensive laboratory resources. However, simulation through Computational FluidDynamics (CFD) software offers inexpensive solutions to aerodynamic blade analysis. Analysisis done in CFD code FLUENT to study the wind velocity effect on blade. A detailed coupledanalysis also performed to predict the structural stability of turbine blade using finite elementanalysis. Structural analysis has done on turbine blade using different materials. The resultsshown with the increase in wind velocity, the variation of the wind turbine's total torquecoefficient tended to be smooth when the rotational speed is constant.The objective of this report describesanalysis of symmetric airfoils of a vertical axis wind turbine using analytical and numerical techniques using FLUENT 15.0.

***IndexTerms* – Wind turbine, wind velocity, CFD.**

I. INTRODUCTION

Whole world is slowly switching to alternative energy sources due to its varied advantages over the limited stock, global warming and health hazards of non-renewable energy sources. Amongst which wind energy is promising with current technology, the low cost of wind energy is competitive with more conventional sources of energy which is abundantly available and more at high altitudes. Energy is a fundamental thing to grow economically and socially. On beginning of 21st era we will face challenging problems to supply energy, energy utilization increasing day to day, we are depending on fossil fuels heavily these are giving pollution threat to climate and also an economic threat [1]. To solve this problem so many developed countries are investing largely in renewable energy resources, as part of the internationalco2 emission decreasing policy Australia was established strategies to produce power from renewable energy of atleast 20 % by the year 2020 [2] .Wind energy is an effective resource to generate power, it is readily available and it is clean, safe. Particularly in Europe it has a share of 70 % of the international wind energy industry [1]. The efforts have been put to analyze the aerodynamic models employed forthe performance of vertical axis wind turbine. The main advantages of Vertical axis wind turbines are quiet, omnidirectional, and create less stress on support structure, self-starting. They require less wind to produce power, the large blades of vertical axis wind turbines with high aspectratio's exposed to a very large value of bending moments due to centrifugal forces, these causes the failure of blades[kragten 2004]. Even small blades of vertical axis wind turbines are dangerous because of the blades spins or rotate very quickly and give acceleration due to lack of stall. The high rotational speeds causes the high centrifugal forces and torque, which normally supports the blades but have a probability of increasing structural failures at that time[Jain 2011].This paper describes the analysis of symmetric airfoils of a vertical axis wind turbine using analytical and numerical techniques and naca0012 airfoil is selected for the same.

II. THE AIRFOIL

An airfoil is a 2D shape capable of producing a reactive lift force when in motion relative to the surrounding air. Figure 1 gives an overview of the basic airfoil terminology.

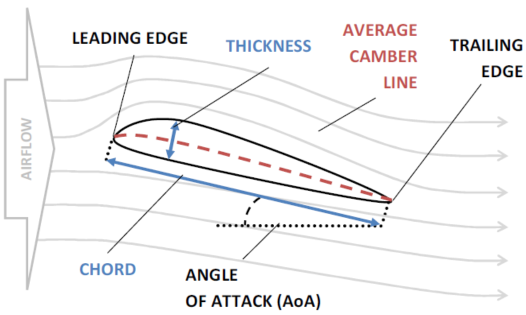


Fig. 1: Airfoil terminology

As the air travels over the top of airfoil, it accelerates and consequently pressure decreases in this area. Lower pressure on the top of the airfoil than the bottom creates a suction force called lift as shown in figure 2. Lift is the force that keeps aeroplanes in the sky. Drag on the other hand, acts in the same direction to the airflow and is generally considered a nuisance, as for example in an aeroplane, extra fuel must be used to overcome the drag forces. Drag forces arise mainly fromfriction between the viscous fluid and the surface of the aerofoil (skin friction drag) and thedifference in pressure between the leading and trailing edges of the aerofoil (form drag).

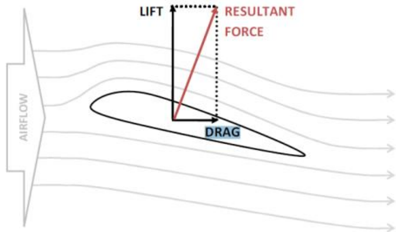


Fig. 2: Lift and drag forces of an airfoil

It is noted that in the wind turbine, the airflow over the aerofoil is not equal to the oncoming wind velocity. Figure 3 illustrates how the airflow onto the aerofoil (the relative wind velocity) is a result of both the oncoming wind velocity and the airflow due to blade rotation.

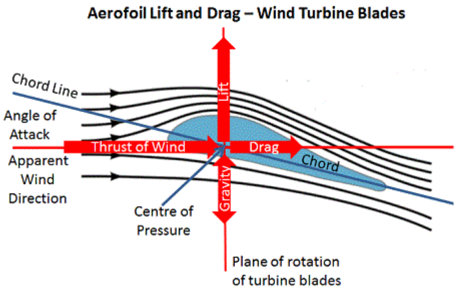


Fig. 3: Lift and drag forces of an airfoil

The airfoils were developed by the national advisory committee for aeronautics. In the naca airfoils four digits describe that

1. The first digit represents max camber as percentage of the chord.
2. The second digit represents a distance of max camber from the airfoil leading edge.
3. The last two digits represent the thickness of the airfoil.

The naca airfoils 0012,0015,0018 and 0025 are most commonly using airfoils profiles are shown in figure 4.

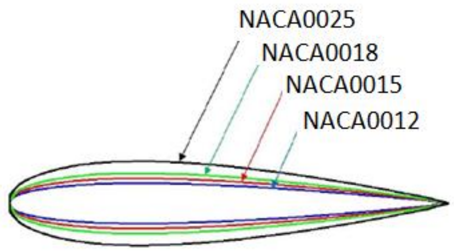


Fig. 4: Profiles of commonly used airfoils

In the present paper, we will discuss about one of the naca symmetric airfoils, that is naca0012.

III. COMPUTATIONAL FLUID DYNAMICS

Governing Equations

The instantaneous equations of mass, momentum and energy conservation can be written as follows in a stationary frame:

- **The continuity Equation:**

$$\frac{\partial y}{\partial x} + \nabla.(\rho U) = 0$$

- **The Momentum Equations**

$$\frac{\partial(\rho U)}{\partial t} + \nabla.(\rho U \otimes U) = -\nabla p + \nabla.\tau + S_M$$

Where the stress tensor, τ, is related to the strain rate by

$$\tau = \mu\left(\nabla U + (\nabla U)^T - \frac{2}{3}\delta\nabla.U\right)$$

- **The Total Energy Equation**

$$\frac{\partial(\rho h_{tot})}{\partial t} - \frac{\partial p}{\partial t} + \nabla.(\rho U h_{tot}) = \nabla.(\lambda\,\nabla T) + \nabla.(U.\tau) + U.S_M + S_E$$

ANSYS Fluent provides comprehensive modeling capabilities for a wide range of incompressible andcompressible, laminar and turbulent fluid flow problems. Steady-state or transient analyses can beperformed.

IV. METHODOLOGY

1. Analytical calculations

The followingare the definitions of various variables used in this model:
E = Kinetic Energy (J)
ρ= Density (kg/m3)
m = Mass (kg)
A = Swept Area (m2)
v = Wind Speed (m/s)
Cp = Power Coefficient
P = Power (W)
r = Radius (m)
dt/dm=Mass flow rate (kg/s)
x = distance (m)
dt/dE=Energy Flow Rate (J/s)
t = time (s)
Under constant acceleration, the kinetic energy of an object having mass m and velocity v is equal to the work done W in displacing that object from rest to a distance s under a force F , i.e.:
E =W = Fs
According to Newton's Law, we have:
F = ma
Hence,
E = mas …..(1)

The power can be defined as: **P = 1/2ρAV^3…(2)**

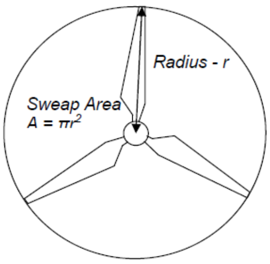
A German physicist Albert Betz concluded in 1919 that no wind turbine can convert more than 59.3% of the kinetic energy of the wind into mechanical energy turning a rotor. To this day, this is known as the Betz Limit or Betz' Law. The theoretical maximum power efficiency of any design of wind turbine is 0.59 (i.e. no more than 59% of the energy carried by the wind can be extracted by a wind turbine). This is called the "power coefficient" and is defined as:
C pmax =0.59. Also, wind turbines cannot operate at this maximum limit. The Cp value is unique to each turbine type and is a function of wind speed that the turbine is operating in. Once we incorporate various engineering requirements of a wind turbine - strength and durability in particular – the real world limit is well below the Betz Limit with values of 0.35-0.45 common even in the best designed wind turbines. By the time we take into account the other factors in a complete wind turbine system - e.g. the gearbox, bearings, generator and so on - only 10-30% of the power of the wind is ever actually converted into usable electricity. Hence, the power coefficient needs to be factored in equation (2)and the extractable power from the wind is given by:2

P= 1/2ρAv3 Cp…(3)

The swept area of the turbine can be calculated from the length of the turbine blades using the equation for the area of a circle:
A = πr 2 …(4)
where the radius is equal to the blade length asshown in the figure below:

We are given the following data:
Blade length, l = 52 m
Wind speed, v = 10 m/sec
Air density, ρ= 1 kg/m3
Power Coefficient, Cp = 0.4

Inserting the value for blade length as the radius of the swept area into equation (4)



P=0.5×1×8495×1000×0.4=1699 KW
From numerical value, V=9.88 m/s
Therefore, P=1699×9.88×9.88×9.88=1638 KW

2.Selection of airfoil

Symmetrical airfoils are used for a small scale VAWT (vertical axis wind turbines), these have similar characteristics of lift and drag on upper and lower surfaces. The advantage is that symmetrical airfoils provide lift from both side of the airfoil, so these will give a lift during 360o rotation and we are not having a problem to adjust the blades relative to wind direction. For vertical axis wind turbines commonly used symmetrical airfoils are naca0012,0015, 0018, and naca0025. The airfoil naca0012 was chosen and analysis of this symmetric airfoil of a vertical axis wind turbine was done using analytical and numerical techniques and validated naca0012 airfoil with experimental data.

3.Geometry and Mesh

Geometry model is created in ANSYS workbench coordinates of blade imported from NACA airfoil library and blade profile generated. For numerical simulations the co-ordinates for all airfoils are obtained from NASA website for simulations. Creation of fluid domain and mesh generated by using ANSYS FLUENT 15.0. The 2D computational mesh designed in C-type rectangular domain, it consists of cells and domain names shown in figure 5 and figure 6.

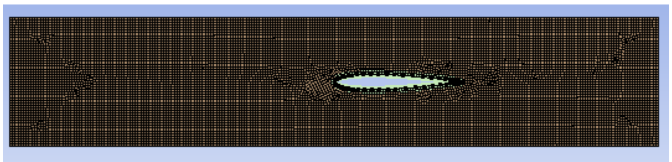


Fig. 5: Mesh model

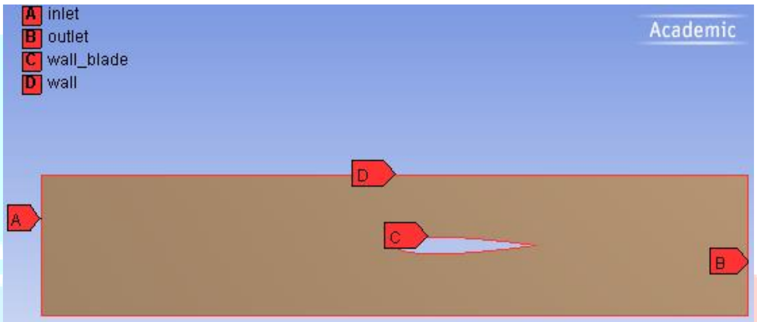


Fig. 6: Boundary names

4. Boundary conditions

Inlet is defines as velocity inlet, simulation is done under a steady wind velocity of 2m/s with wind angle of attack on blade is varied 0^o and 10^o.

V. RESULTS AND DISCUSSION

1. Contours of pressure magnitude

The contours of pressure magnitude obtained for various angles of attack from CFD simulations are shown in the following Fig.7, 8, 9and 10. We can see the flow accelerates on the upper side of the airfoil and the velocity of flow decreases along the lower side, according to Bernoulli's principle the upper surface will experience low pressure and the lower surface will experience higher pressure. Hence the value of lift coefficient will increase and the value of drag coefficient will also increase but the increase in drag is low in comparison to the increase in lift force.As the pressure on the lower surface of the airfoil is greater than that of the incoming flow stream, the airfoil is effectively pushed upward normal to the incoming flow stream.

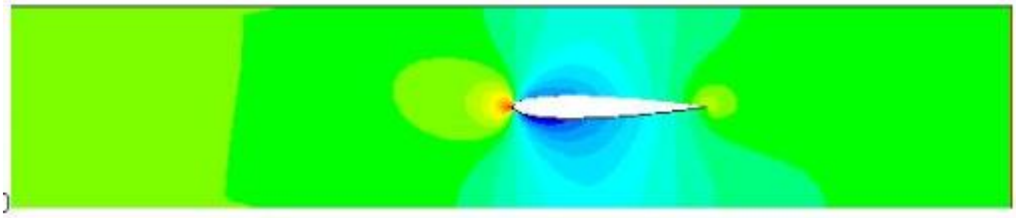


Fig.7: Static pressure at 0^0 angle of attack, v=2m/s

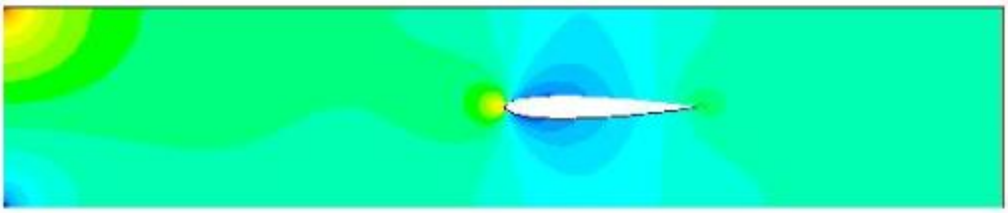


Fig. 8: Static pressure at 10^0 angle of attack, v=2m/s

2. Contours of velocity magnitude

The contours of velocity magnitude obtained for various angles of attack from CFD simulations are shown in the following Figures. 3, 4, 5 and 6. On the leading edge, we can see the stagnation point where the velocity of flow is nearly zero. The fluid accelerates on the upper surface of the airfoil while the velocity of the fluid decreases along the lower surface of the airfoil.

Fig. 11: Velocity contours at 0^0 angle of attack, v=2m/s

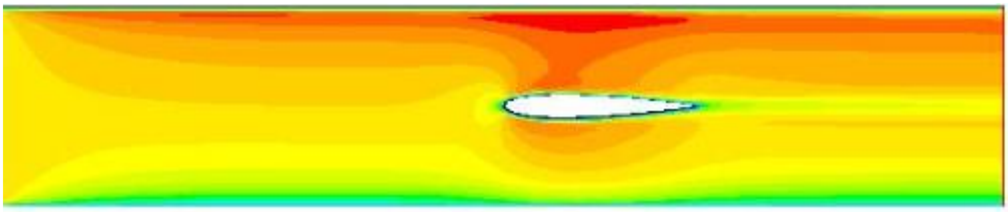


Fig. 12: Velocity contours at 10^0 angle of attack, v=2m/s

The Experimental data used for validation of NACA 0012, the flow properties are considered here are M=0.15, Re = 6 million validation [Abbott. 1959], [7]. The variation of coefficient of lift (CL) with angle of attack (α) is compared with the experimental results as shown in figure11. The numerical results are good agreement with experimental results as shown in figure11

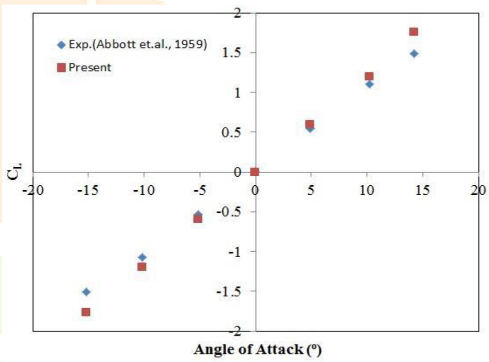


Fig. 11: Coefficient of lift with respect to Angle of attack

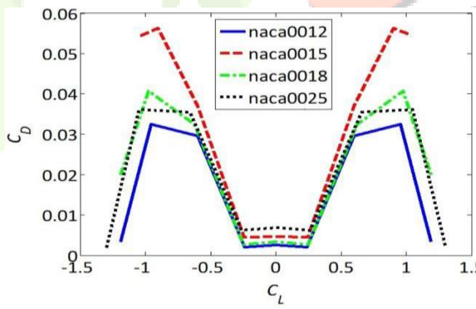


Fig. 12: Coefficient of lift and Coefficient of drag for various blade profiles

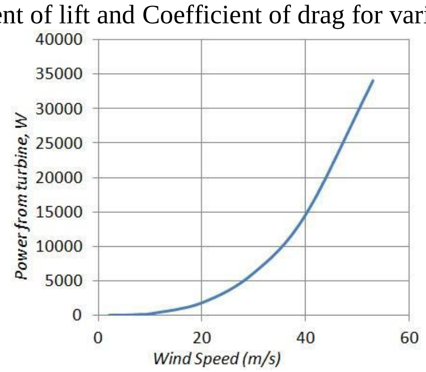


Fig. 13: Power of turbine with wind speed

VI. CONCLUSIONS

With the help of CFD software Ansys-Fluent, successful analysis of aerodynamic performance of NACA0012 airfoil has been carried at various angles of attack (0, 4, 6, 10 degree) with constant Reynolds number (106)using the Realizable $k - \varepsilon$turbulence model. The following conclusions were made from the simulation

- It is seen that the velocity of upper surface is higher than the velocity of the lower surface.
- The pressure coefficient of the airfoil's upper surface was negative and the lower surface was positive, thus the lift force of the airfoil is in the upward direction.
- The coefficient of pressure difference is much larger on the front edge, while on the rear edge it was much lower.
- It is observed that to increase the value of lift force and lift coefficient we have to increase the value of angle of attack. This leads to rise in drag force and drag coefficient as well, but the increase in drag force and drag coefficient is quite low in comparison to lift force and lift coefficient.

Computed drag and lift forces were found in close agreement with the experimental values thus corroborating that CFD analysis is an efficient alternative to experimental methods.

VII. ACKNOWLEDGMENT

I would like to give thanks to our HOD Dr.P.Ravinder Reddy for his acceptancy to do simulation in department CFD. It is my pleasure to give thanks to Dr.M.V.S.MuraliKrishna coordinator M.E thermal engineering who helped us to define the problem.

REFERENCES

[1] TonioSant, 2007. BEM-based aerodynamic models in wind turbine design codes, university of Malta.
[2] Abdulkadir Ali, Steve Golde, FirozAlam, HazimMoria, 2012.Experimental and computational study of micro vertical axis wind turbine, Australia.
[3] Kragten. A, 2004. The Darrieus Rotor, a vertical axis wind turbine report KD215. Engineering office kragten design, populierenlaan 51, 5492 Sint – Oedenrode.
[4] Jain, Pramod, 2011. Wind energy engineering. McGraw Hill Companies, ISBN 978-0-07-171478 -5MHID: 0-07-171478 -2,
[5] V. Esfahanian, A. Salavati Pour, I. Harsini, A. Haghani, R.Pasandeh, A. Shahbazi, G. Ahmadi, 2013. Numerical analysis of flow field around NREL PhaseII wind turbine by a hybrid CFD/BEM method.
[6] M. Ragheb, 2013. Vertical axis wind turbine. Ali, A. 2001.Macroeconomic variables as common pervasive risk factors and the empirical content of the Arbitrage Pricing Theory. Journal of Empirical finance, 5(3): 221–240.
[7] W. J. McCroskey, **A critical assessment of wind tunnel results for the NACA 0012 airfoil,** Aeroflightdynamics Directorate, U.S. Army Aviation Research and Technology Activity, Ames Research Center, Moffett Field, California
[8] B. Sanderse, S.P. van der Pijl, B. Koren, **Review of computational fluid dynamics for wind turbine wake aerodynamics, 10.1002/we.458, wind energy, 2011**
[9] Muhammad MahmoodAslam BhuttaNasirHayatAhmed UzairFarooqZainAliSh. RehanJamilZahidHussain, Vertical axis wind turbine – A review of various configurations and design techniques, elsavier, Renewable and Sustainable Energy Reviews Volume 16, Issue 4, May 2012, Pages 1926-1939
[10] Andrea Alaimo *, Antonio Esposito, Antonio Messineo, Calogero Orlando and Davide Tumino, 3D CFD Analysis of a Vertical Axis Wind Turbine, *Energies* 2015, *8*(4), 3013-3033; doi:10.3390/en8043013,