Aerodynamic Analysis of Wind Turbine Rotor Blade Airfoils by CFD Simulation

[1]Salunkhe Shekhar Prakash, [2]Prof. Patil Sandeep Madhavrao
[1]Reaserch Student, Mechanical Engineering, SGDCOE, Jalgaon, Maharashtra, India
[2]Head of Department, Assistant professor, Mechanical Engineering, SGDCOE, Jalgaon, Maharashtra, India

ABSTRACT

One of the major challenges in the new century is the efficient use of energy resource as well as production of energy from the renewable energy resources. Although, scientist from around the world have shown that global warming has been caused in part by the greenhouse effect which is largely due to the use of fossil fuel for production of electricity and transportation facility. There are several alternative forms of energy that have already been developed such as wind, solar, geothermal, tidal, and hydroelectric power. The advancement in renewable energy technologies has been possible and the vast amount of research performed by engineer in order to make them more affordable, more efficient, most importantly and inexpensive. Wind energy is one of the most important sources of renewable energy. Wind turbine extract energy from the kinetic energy of the wind. At present many researches are concentrated on aerodynamic design of wind turbine rotor blade through wind tunnel test and blade element momentum. These conventional methods are difficult and time consuming. However, wind turbine blade simulation through computational fluid dynamics (CFD) offers less time consuming, easy and inexpensive way to aerodynamic blade design. In this study two dimensional airfoils (NACA 4424 and NREL S809) CFD models are present using ANSYS-FLUENT software. Using the Spalart-Allmaras viscosity the dimensionless lift and drag coefficient and forces are calculated using different angle of attack and different mach numbers. Viscosity is based on Sutherland model. One of airfoil is selected from the different two airfoils, which is responsible for maximum power and efficiency. Wind data is taken from the review paper based on wind speed of Bhopal bairagarh site.

KEYWORDS

The research aims to evaluate the aerodynamic performance of variable speed fixed –pitch horizontal axis wind turbine blades airfoils through two-dimensional computational fluid dynamics (CFD) analysis.

The objective of the research is to establish two dimensional CFD model of wind turbine blade airfoil-

1) To analyses the aerodynamic performance of different airfoils at two different wind velocity.
2) Compare different airfoil according to lift and drag coefficient.
3) Select one airfoil which is most efficient accordingly local site.

1 INTRODUCTION

1.1 Background

Energy is essential for civilisation development. India is developing country and with economic and socialisation progress, there is an expanding demand of renewable energy to secure the conventional energy source for long period. As a clean renewable energy source, wind energy plays very important role in modern life. Wind energy is an abundant resource as comparison to other renewable energy resource. Moreover unlike solar power usage does not depend on weather and climate condition. Wind turbine was invented to extract energy from the wind. Power in the wind turbine comes from transformation of air that is driven from the heat of the sun, which is abundant, clean and renewable. As one of the most popular energy source, wind energy exploitation is growing rapidly.

The development of wind power in India began in the 1986 with first wind farms being set up in the coastal areas of Maharashtra (Ratnagiri), Gujrat (Okha) and Tamilnadu (Tuticorin) with 55 kW Vestas wind turbines. These demonstration projects were supported by MNRE. The capacity has significantly increased in the last few years. Although a relative newcomer to the wind industry compare to Denmark or the United State, India has the fourth largest installed wind power capacity in the world. In 2009-10 India's growth rate was highest among the other top four countries.

As of 31th march 2016 the installed capacity of wind power in India was 26,769 MW, mainly spreads across south, west and north region. East and north-east regions have no grid connected wind power plant as of march, 2015 end. In the year 2015 the MNRE set the target for wind power generation capacity by the year 2022 at 60,000 MW. The worldwide installed capacity of wind power reached 435 GW by the end of 2015. China (148,000 MW), US(74,347 MW) and Germany (45,192 MW) are ahead of India in fourth position. The short gestation periods for installing wind turbine, and the increasing reliability and performance of wind energy machines has made wind power a favourable choice for capacity addition in India. Wind power accounts nearly 8.6% of India's total installed power generation capacity and generated 28,604 million Kwh (MU) in the fiscal year 2015-16 which is nearly 2.5% of total electricity generation. The capacity utilisation factor is nearly 14% in the fiscal year 2015-16 (15% in 2014-15). 70% of wind generation is during the five months duration from May to September coinciding with southwest monsoon duration.

Installed Wind Power Capacity

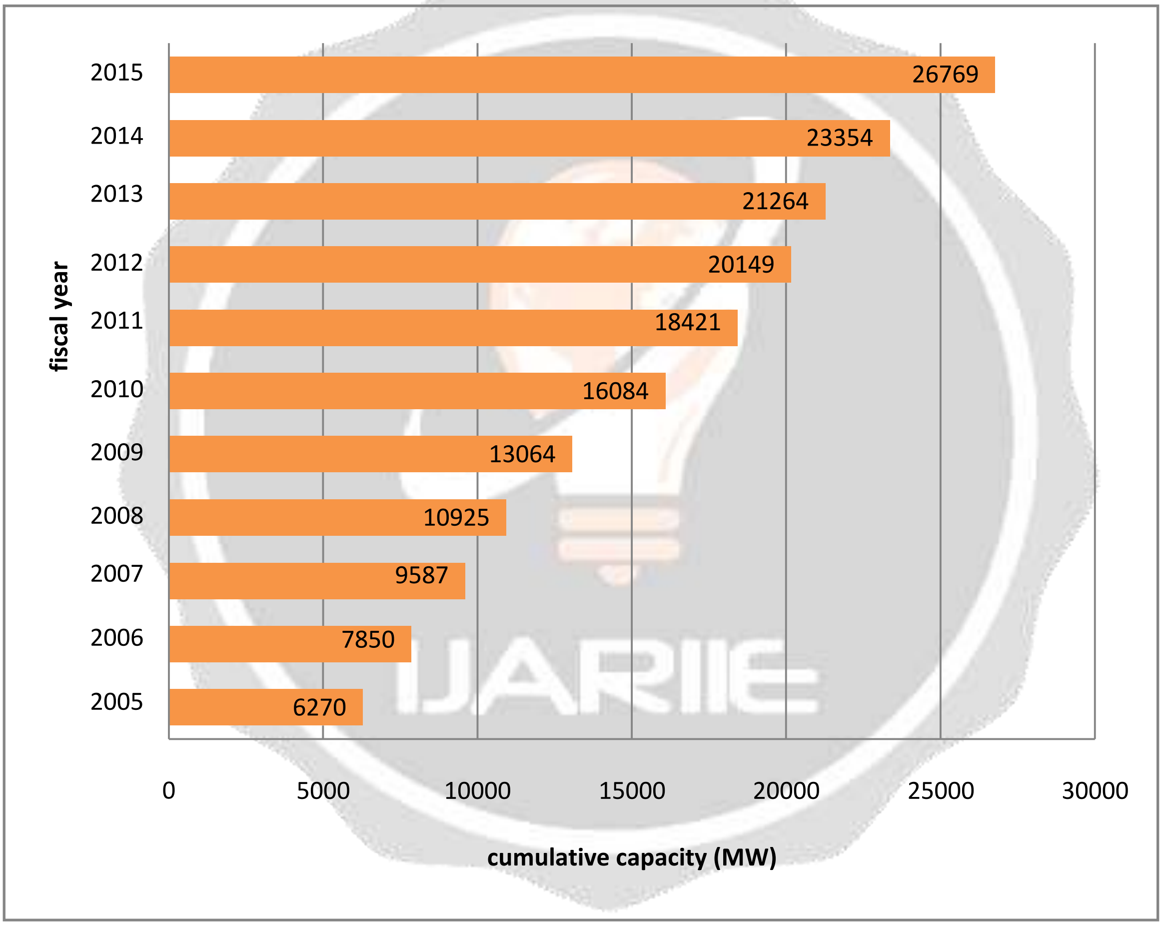

Figure 1.1 Installed wind power capacity in India

A wind turbine consist of several parts i.e. blade, rotor, generator, control system, driven chain etc. The rotor is driven by the wind and rotates at predefined speed by the virtue of the wind speed, so that the generator can produce electricity under the regulation of the control system. In order to get the maximum kinetic energy from the wind, engineers put much efforts on the design of effective blade geometry. In the early stage, the aerofoil of helicopters were used for wind turbine blade airfoil design but now many specialize aerofoil have been invented and used for wind turbine blade design. A rotor blade has different airfoil at different section to improve the efficiency, so the present time blades are more complicated and efficient than the early age wind turbine blades.

In the early stage, the research on wind turbine rotor blade was limited on theoretical only, field testing and wind tunnel testing which need a lot of effort and resource. Because of development of computer aided design codes, they provide another way to design and analyse the wind turbine blades. Aerodynamic design of wind turbine blade can be analyse using computational fluid dynamics (CFD), which is one of the branch of fluid mechanics that uses numerical method and algorithms to solve and analyze problem of fluid flow. Comparing to conventional theoretical and experimental method, numerical method saves money and time for the performance analysis and optimal design of wind turbine rotor blade.

A wind turbine converts kinetic energy into mechanical power through a rotor, and then converts the mechanical power into electrical power through a generator which is linked to the rotor with gearbox. Various type of wind turbine are designed to get advantage of wind energy based on the principle of aerodynamics. Depending upon the wind turbine rotor orientation, there are two type of wind turbines, horizontal axis wind turbine (HAWT) and vertical axis wind turbine (VAWT). Generally according to wind turbine capacity, modern wind turbine can be classified as small wind turbine (below 50kW), medium wind turbine(50kW-250kW) and large wind turbine(above 250kW). When considering installation sites, there are onshore and offshore wind turbines. Based on operation scheme, wind turbine can be divided into fixed pitch wind turbine and variable pitch wind turbine. According to the relative flow direction of the wind turbine rotor, HAWT are either upwind or downwind turbines. Most modern horizontal axis wind turbine have three blade, however there are also turbine with two blade. For small wind turbine, there are turbine with 5 or 7 blade also. Three bladed horizontal axis wind turbine is the most common topology due to higher efficiency, better controlled performance and aesthetic appreciation.

2 Literature Review

In this some basic theories of wind turbine aerodynamics and computational fluid dynamics are introduced. Moreover the purpose and methods for wind turbine simulation are discussed.

2.1 Historical development of wind turbine

Wind turbine is a device, which converts the kinetic energy from the wind to electrical energy by a mechanical rotor, a drive train and a generator. One of the earliest wind turbine was designed by Poul La Cour, who was a professor at an adult education centre in Denmark in 1981. Nowadays Enercon E-126, the world biggest wind turbine can generate up to 7 MW of power under the rated wind speed. This capacity can provide the daily electricity for more than 4500 homes. Following the technology development of modern wind turbine, they can now be mounted either on the ground or on the seabed. A giant offshore wind turbine of 10 MW will be installed in 2011 by Enova SF in Norway. As the depletion of coal and fossil fuel, wind energy plays an important role in this century.

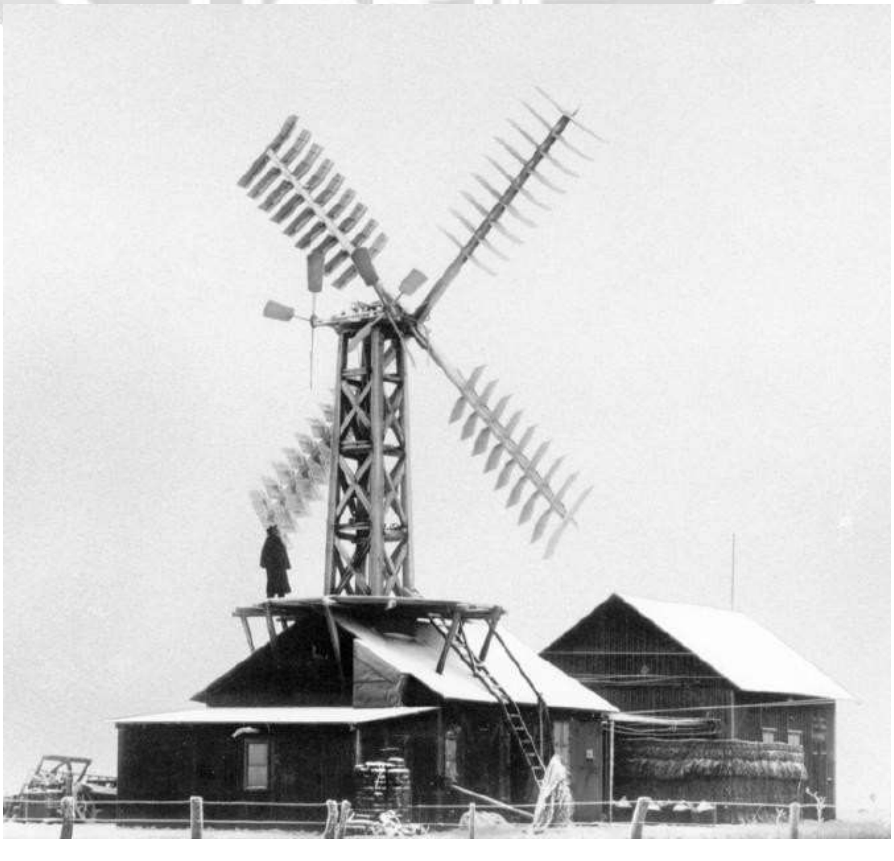

Figure 2.1 Pou La Cour's first electricity producing wind turbine in 1891 in Askov, Denmark(Golding 1977)

2.2 Wind turbine aerodynamics

According to the different rotational orientation, wind turbine can be categories as vertical axis wind turbine and horizontal axis wind turbine. The advantage of vertical axis wind turbine are-

1) Simple structure: vertical axis wind turbine can work without yaw system and most of them have a blade with constant chord and no twist, which is easy to construct.
2) Easy to install: because the drive trains can be located relative to the ground.

Comparing to horizontal axis wind turbine, stall control can only be used in vertical axis wind turbine as it is difficult to incorporate aerodynamics control such as variable pitch and aerodynamic brake, so the overall power efficiency is lower than horizontal axis wind turbine.

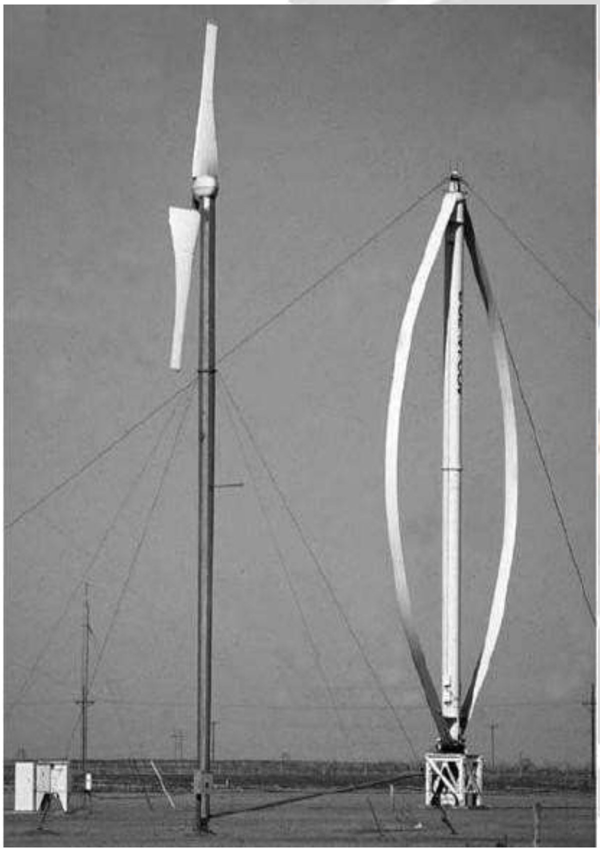

figure 2.2 Vertical axis wind turbine

figure 2.2 Horizontall axis wind turbine

2.3 Computational fluid dynamics (CFD)

There are many commercial CFD software used in engineering, such as PHOENICS (it is the first commercial CFD software), STAR-CD, ANSYS FLUENT/CFX and so on. All CFD software have three main structures which are Pre-Processer, Solver and Post-Processor.

2.4 Issue in wind turbine simulation using CFD software

A confident result of airfoil simulation was achieved in two dimensional simulation, but it was difficult to get a reliable result for three dimensional simulation. Initially the air flow passing through a rotating horizontal axis wind turbine blade is much more complicated than that of a two dimensional simulation because the changing angle of attack vary along the airfoil span. Moreover under high wind speed stall of the system can take place from the root section. There also centrifugal force along the blade due to rotor rotation. On the other hand accuracy of simulation is affected because of the limitation of CFD software. Firstly no matter what kind of turbulent model is used, it is extremely hard to simulate the turbulence in physical reality. Additionally fine mesh is a prerequisite in order to simulate full scale wind turbine, which are very memory restricted inside the computer meaning the simulation cannot be carried out using personal computer with low configuration. In order to reduce the mesh size normally, neither the simulation nor the ground are included into the model. Finally geometry of wind turbine blade is difficult to mesh with quality. Most wind turbine blade tip are designed using a thin airfoil for low induced drag and the root region is using a thick version for structural support, the size different between tip and root leads to mesh scales difficult to control.

3 Methods

GRAMBIT is the pre-processor of ANSYS-Fluent. A simulation model can be created directly in GRAMBIT or import from the other CAD software such as SolidWorks and Pro/Engineer®. In this section airfoils NACA 4424 and NREL S809 are modelled.

4 Results and Discussions

In this aerodynamic analysis of airfoils will be conducted in two-dimensional simulation using ANSYS Fluent. NACA 4424 and NREL S809 airfoils will be compare in order to find out which one has a better aerodynamic performance.

5 Future Work

For CFD method analysis, software like ANSYS with CFX should be linked with PRO/E. So an actual lifts and drag reduction, vortex effects for the respective winglet can be calculated by varying the parameters. Optimization of the geometry for each type of winglet could be done, by targeting drag as minimizing factor.

6 References

1 Siddharth Joshi, “Design, Simulation And Analysis Of Grid Connected Wind Energy Conversion System”, A Thesis Submitted to GTU the Degree of Master of Engineering in Power System, June 2011 PP14.

2 G.M. Joselin Herbert et al, “A review of wind energy technologies”, Renewable and Sustainable Energy Reviews 11 (2007) PP 1117–1145.

3 Wikipedia, the free encyclopedia, http://en.wikipedia.org/wiki/wind_power

4 Metin Ozen, Ashok Das, Kim Parnell, “CFD Fundamental and Applications”, PP 7-8.

5 Prashant Bhatt et al, “Computational Fluid Dynamics Analysis of Wind Turbine Rotor Blades- A Review”,IJCRR, Nov 2012 / Vol 04 (21) Page 163.

6 H. V. Mahawadiwar, V.D. Dhopte, P.S.Thakare, Dr. R. D. Askhedkar, “CFD Analysis of Wind Turbine Blade”, International Journal of Engineering Research and Applications, May-Jun 2012, PP- 3188-3194.

7 Ji Yao, Jianliang Wang, Weibin-Yuan, Huimin Wang, Liang Cao, “Analysis on the Influence of Turbulence Model Changes to Aerodynamic Performance of Vertical Axis Wind Turbine”, ELSEVIER, International Conference on Advances in Computational Modelling and Simulation, Procedia Engineering 31 (2012) 274-281.

8 Chris Kaminsky, Austin Filush, Paul Kasprzak and Wael Mokhtar, “A CFD Study of Wind Turbine Aerodynamics”, Proceedings of the 2012 ASEE North Central Section Conference.

9 C. Rajendran, G. Madhu, P.S. Tide, K. Kanthavel, “Aerodynamic Performance Analysis of HAWT Using CFD Technique”, European Journal of Scientific Research, ISSN 1450-216X Vol. 65, No. 1 (2011), PP 28-37.

10 R.S. Amano, R.J. Malloy, “CFD Analysis on Aerodynamic Design Optimization of wind Turbine Rotor Blades” , World Academy of Science and Technology60 2009.

11 Le Pape A and Lecanu J, “3d navier-stokes computations of a stall-regulated wind turbine”, Wind Energy,7(4):309–324, October-December 2004.

12 Piggott H. “Small Wind Turbine Design Notes” http://users.aber.ac.uk/iri/WIND/TECH/WPcourse/index.html