International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 1 Issue 10, December- 2012

A Review on Aerodynamic Analysis of Wind Turbine Blade Using CFD Technique

Mr. Bhargav Patel
Research scholar

Prof. D. A. Patel
Associate Professor

Department of Mechanical Engineering, Sankalchand Patel College of Engineering,
Gujarat Technological University, Visnagar-384 315, Dist: Mehsana, Gujarat, India

ABSTRACT

The aim of this paper is review for various researches on the aerodynamic analysis of wind turbine blade using Computational Fluid Dynamics (CFD). Wind turbine blades are subjected to various aerodynamic loads; currently much research has concentrated on improving the aerodynamic performance of wind turbine blade through wind tunnel testing and theoretical studies. These efforts are much time consuming and need expensive laboratory resources. However, wind turbine simulation through Computational Fluid Dynamics (CFD) software offers inexpensive solutions to aerodynamic blade analysis problem. In this study the researchers used various CFD codes and got a good result through these codes.

Keywords: Wind Turbine blade, Computational Fluid Dynamics (CFD), National Renewable Energy Laboratory (NREL)

1. INTRODUCTION:

A wind turbine is a rotating machine, which converts the kinetic energy in wind into mechanical energy. If the mechanical energy is then converted to electricity, the machine is called a wind generator, wind turbine, wind power unit (WPU), wind energy converter (WEC), or aero generator. Wind turbines can be separated into two types based by the axis in which the turbine rotates. Turbines that rotate around a horizontal axis are more common. Vertical axis turbines are less frequently used [1]. Blades are the most critical component of a wind turbine which is responsible for conversion of wind energy into kinetic energy. Aerodynamics is a science and study of physical laws of the behavior of objects in airflow and the forces that are produced by airflows [2]. In the early stage, the research on wind turbine blade design was limited on theoretical study, field testing and wind tunnel testing which need a lot of efforts and resources. Due to the development of computer aided design codes, they provide another way to design and analyzed the wind turbine blades. Aerodynamic performance of wind turbine blades can be analyzed using computational fluid dynamics (CFD), which is one of the branches of fluid mechanics.

2. COMPUTATIONAL FLUID DYNAMICS:

Computational Fluid Dynamics (CFD), which is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems of fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary condition [3]. Computational Fluid Dynamics (CFD) has grown from a mathematical curiosity to become an essential tool in almost every branch of fluid dynamics, from aerospace propulsion to weather prediction. CFD is commonly accepted as referring to the board topic encompassing the numerical solution, by computational methods, of the governing equations which describe fluid flow, the set of the Navier-Stokes equations, continuity and any additional conservation equations, for example energy or species concentrations [4]. As a developing science, Computational Fluid Dynamics has received extensive attention throughout the international community since the advent of the digital computer. All CFD analysis starts by defining the geometry to be used. However, for CFD, the geometry is the geometry where the air (or fluid) will flow. This means that it is often necessary to define artificial boundaries for inlets, outlets, far-field conditions etc. CFD provides five major advantages compared with experimental fluid dynamics [5].

- Lead time in design and development is significantly reduced.
- CFD can simulate flow conditions not reproducible in experimental model test.
- CFD provides more detailed and comprehensive information.
- CFD is increasingly more cost-effective than wind tunnel testing.
- CFD produces lower energy consumption.

A well developed CFD code allows alternative designs to be run over a range of parameter values e.g. Reynolds number, Mach number, flow orientation [5].

3. LITERATURE REVIEW:

Literature review is one of the scope studies. It works as guide to run this analysis. It will give part in order to get the information about wind turbine blade analysis using CFD. From the early stage of the project, various literature studies have been done. Research journals, books, printed or online conference article were the main source in the project guides.

H. V. Mahawadiwar et al (2012) carried out Computational Fluid Dynamics (CFD) analysis of wind turbine blade with complete drawing and details of sub-system. The blade material is Cedar wood, strong and light weight. CAD model of the blade profile using Pro-E software is created and the flow analysis of the wind turbine blade mesh is created in the GAMBIT software. CFD analysis of the wind turbine blade is carried out in the FLUENT software. Form this study they conclude as following:

1. Value of numerical power increases as angle of attack increases from 0^0 to 7^0, after 7^0 the value of numerical power reduced. Hence critical angle of attack for this blade is 7^0.
2. The maximum value of coefficient of performance (CP_{max} = 0.271) was saw at angle of attack 7^0 and at velocity of air 8 m/s.
3. This blade can generate maximum power of 620 W at maximum CP, angle of attack 7^0 and velocity of air 8 m/s.
4. From the graph Fig 3.1 they was observed that coefficient of performance is increases from 3 m/s to 8 m/s and after 8 m/s value of coefficient of performance reduced [6].

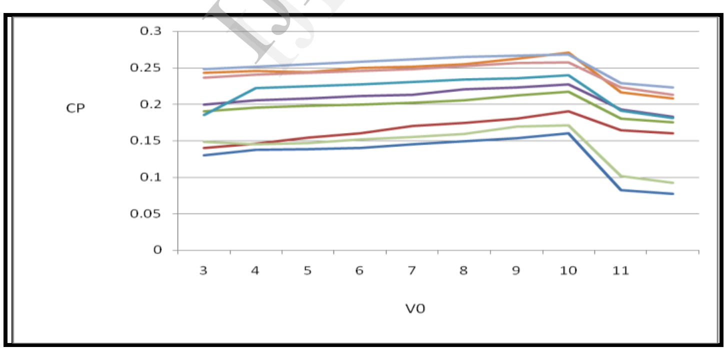

Fig 3.1 Graph of CP versus V_0

Ji Yao et al (2012) creates the 2D model for the unsteady numerical simulation of VAWT by FLUENT software and algorithm SIMPLEC combined with the sliding grid technology. For numerical simulation it used two turbulence models. They use NACA 0018 airfoil series for created the blade 2D model. Domain C-H type for CFD analysis C is a half semicircle shape whose radius = 16m & H is a rectangle whose size are = 32m * 30m. In FLUENT they used various parameters for analysis are Pressure, Velocity, Turbulent Kinetic Energy, Pressure Velocity Coupling and Grid Generation. They use Standard □-ε, RNG □-ε as Turbulence Model. The results showed that the influence of different turbulence models on the velocity field is less, on the pressure field is relatively large, and on the value of the total torque is much larger [7].

Chris Kaminsky, Austin Filush, Paul Kasprzak and Wael Mokhtar (2012) carried out the research of a VAWT using the NAC A0012-34 airfoil. The system was modeled in Solid Works. They are use of the STAR CCM

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 1 Issue 10, December- 2012

software to CFD analyzes the air flow around a vertical axis wind turbine to perform. Analysis has been done in three ways as show:

1. To determine CFD analysis analyzed the 2D flow over the chosen airfoil.
2. Determine the analysis looked at the flow over a 3D representation of the airfoil.
3. Finally, a full VAWT assembly was created and analyzed at various wind directions at the same wind speeds.

The airfoil then the 2D and 3D simulations used different angles of attack (0 to 15 degrees) and speeds (15 & 30 mph) to determine. The full assembly included 3 airfoils that were attached into a 5ft high, 3 ft diameter structure. The results of this research on the NACA 001234 airfoil showed it could be a very viable choice for a residential VAWT. The 2D analysis gave a stall angle of about 8 degrees, however, the 3D analysis, it being more accurate, did not provide us with a stall angle. The results for the 3D full assembly analysis of vertical axis wind turbine were incomplete [8].

C. Rajendran, G. Madhu, P. S. Tide et al (2011) had demonstrates the potential of an incompressible Navier–Stokes CFD method for the analysis of horizontal axis wind turbines. The CFD results are validated against experimental data of the NREL power performance testing activities.
Comparisons are shown for the surface pressure distributions at several conditions are show as under taken:

a) Wind Velocity is 12.5m/s.
b) Yaw Angle is 0°.
c) Rotational Speed is 25 rpm.
d) Turbulence Model is □-ω SST [9].

R.S. Amano, R.J. Malloy (2009) had study on the optimization of aerodynamic design of wind turbine rotor blade by CFD with two cases one is Straight edge blade and second Swept edge blade. In this work they explore the possibility of increasing the efficiency of the blades at higher wind speeds while maintaining efficiency at the lower wind speeds. They construct different domain shapes for accomplish the analysis. For analysis it uses different conditions in CFD solvers use as under show:

1. Wind Velocity is 5 m/s to 25 m/s.
2. Reference Frame is moving.
3. Turbulence Model is k- ω use.

With these conditions obtain a pressure and pressure contours. And they concluded pressure contours in swept edge blade is better than straight edge blade. Swept edge Blade produces better power with higher wind speed [10].

David Hartwanger et al (2008) has aims to develop a practical engineering methodology for the CFD-based assessment of multiple turbine installations. They are constructs the 2D experimental model of wind turbine which is of NREL S809 aerofoil series and compared. Their results with 3D CFD model in XFoil 6.3 codes and two ANSYS CFX 11.0 versions. It creates the cylindrical domain whose radius 2L and length 5L where L = turbine radius. For grid generation uses ICEM-CFD (ANSYS) software. In analysis it use □- ω turbulence model. There are two main aims for doing analysis is as under show:

- ✓ The primary aim is to predict the lift and drag for 2D experimental wind turbine.
- ✓ Its secondary aim is to compare the results of Lower CFD Fidelity to Higher CFD Fidelity model.

These two aims fulfill with one boundary condition which is use pressure as an inlet condition. The validation of CFD against 2D blade sections showed that the CFD and XFOIL panel code over-predict peak lift and tend to underestimate stalled flow. The 3D results compared well with experiment over four operating conditions. Results from the 3D corresponding calculated torque output showed good agreement with the 3D CFD model and experimental data. However, for high wind cases the actuator model tended to diverge from the CFD results and experiment [11].

J Laursen, P Enevoldsen, S Hjert (2007) had presented 3D CFD rotor computations of a Siemens SWT-2.3-93 variable speed wind turbine with 45m blades. For CFD analysis they use ANSYS CFX 10.0 and 11.0 solvers. For this, they use incompressible Reynolds Average Navier-Stokes equation (RANS) and SST turbulence model. Creates a computational domain for accomplish the analysis. Obtain a various software results at various wind velocity and radial position as input parameter use for modeled blades are compared with the results from the fully turbulent setup [13].

E Ferrer and X Munduate (2007) have analyzed the effect of wind turbine blade tip geometry numerically using Computational Fluid Dynamics (CFD). Researcher take three different rotating blade tips are compared for attached flow conditions and the flow physics around the geometries are analyzed. For analysis they use FLUENT 6.2 version with □-ω SST turbulence model. They got pressure coefficient, thrust and torque for 3 tips with rotational speed 71.9 rpm and wind speed 7 m/s, 8.5 m/s. It results from the comparison that a better tip shape that produced better torque to thrust ratios in both forces and moments is a geometry that has the end tip at the pitch axis. The work here presented shows that CFD may prove to be useful to complement 2D based methods on the design of new wind turbine blade tips [12].

A. Le Pape and J. Lecanu (2004) has work done on 3D Navier-Stokes computations of a stall-regulated wind turbine developed at ONERA software in the field of CFD simulations of the flow field around wind turbine blades. The compressible a NavierStokes solver has been used to compute 2D and 3D configurations of a 2-bladed wind turbine. Prediction of the S809 airfoil performance is discussed and turbulence model is □-ω SST. 3D computations are then presented, analyzed, and compared to the experimental results [14].

4. CONCLUSION:

From the above reviews we can conclude that a number of researchers are using CFD to study wind turbine aerodynamic analysis of blade. CFD is widely used for calculated the flow analysis around the wind turbine rotor blade (e.g. velocity distribution, pressure distribution etc.) which is affected by changing wind velocity, angle of attack, tip speed ratio etc. CFD has become a mature tool for predicting a wide range of flows; however, one important ongoing challenge is the accurate representation of turbulence.

5. 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 PP 14.

[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 Modeling 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 Technology, 60 2009.

[11] David Hartwanger and Dr. Andrej Howat, "3D Modeling of a Wind Turbine Using CFD", NAFEMS Conference, 2008.

[12] E Ferrer and X Munduate, "Wind Turbine Blade Tip Comparison Using CFD", Journal of Physics: Conference Series 75 (2007) 012005.

[13] J Laursen, P Enevoldsen, S Hjert, "3D CFD Quantification of the Performance of a Multi- Megawatt Wind Turbine", Journal of Physics: Conference Series 75 (2007) 012007.

[14] 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.