IJCRR
Vol 04 issue 21
Section: Technology
Category: Review
Received on: 06/08/12
Revised on: 19/08/12
Accepted on: 01/09/12

COMPUTATIONAL FLUID DYNAMICS ANALYSIS OF WIND TURBINE ROTOR BLADES- A REVIEW

Prashant Bhatt, Rajesh Satankar

Department of Mechanical Engineering, Jabalpur Engineering college, Jabalpur

E-mail of Corresponding Author: prashantindore11@gmail.com

ABSTRACT

This review paper presents various researches on the aerodynamic analysis of wind turbine rotor blade through CFD (computational fluid dynamics). An aerodynamic analysis CFD method is good, less expensive and gives the best results. CFD results give good agreement with the experimental data. The wind turbine blades are subjected to various aerodynamic loads, which simulated easily through CFD analysis because experimentally calculation is very complicated. In this study the researchers used various CFD codes which are based on Blade Element Momentum Theory (BEM) and got a good result through these codes.
Keywords -: Wind Turbine blade, Computational Fluid Dynamics (CFD)

INTRODUCTION

The wind is a free, clean, and inexhaustible energy source. It has served mankind well for many centuries by propelling ships and driving wind turbines to grind grain and pump water [1]. A wind turbine is a device that exploits the wind's kinetic energy by converting it into useful mechanical energy. It basically consists of rotating aerodynamically surfaces (blades) mounted on a hub/shaft assembly, which transmits the produced mechanical power to the selected energy utilize (e.g. milling or grinding, pump or generators) [2]. The first use of wind power was to sail ships in the Nile some 5000 years ago. The Europeans used it to grind grains and pump water in the 1700s and 1800s. The first windmill to generate electricity in the rural U.S.A. was installed in 1890 [3]. There are of two types of wind turbine [2]:

a. Horizontal Axis Wind Turbine (HAWT)**.**

b. Vertical Axis Wind Turbine (VAWT)

COMPUTATIONAL FLUID DYNAMICS

Computational Fluid Dynamics indicates the numerical solution of differential governing equations of fluid flows, with the help of computers. This technique has a wide range of engineering applications. In the field of aerodynamic research this technique has become increasingly important and it is prominent for studying turbo-machinery [4]. Computational Fluid Dynamics also provides the convenience of being able to switch off specific terms of governing equations. This permits the testing of theoretical models and, inverting the connection, suggesting new paths for theoretical explorations. CFD provides five major advantages compared with experimental fluid dynamics:

a. Lead time in design and development is significantly reduced.
b. CFD can simulate flow conditions not reproducible in experimental model test.
c. CFD provides more detailed and comprehensive information.
d. CFD is increasingly more cost-effective than wind tunnel testing.
e. 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]. The most important reason for the growth of CFD is that for much mainstream simulation, CFD is significantly cheaper than wind tunnel testing and will become even more in future [6].

LITERATURE REVIEW

Nilay Sezer-Uzol* (2006) -: Author got their result of 3D and time accurate CFD simulation of horizontal axis wind turbine blade by PUMA 2 software. They got result with three different cases:

Case Condition	Wind Velocity	Yaw Angle
Pre Stall	7m/s	0°
Pre Stall Yawed	7m/s	30°
Pre Stall	15m/s	0°

Also obtain pressure coefficients and compare the result with experimental data of NREL [7].

E Ferrer (2007) -: Researcher investigated CFD analysis of three different blade tip comparison with attached flow condition. For analysis they use FLUENT 6.2 version with k-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. Their result shows better tip shape to produce better torque to trust ratio and validate their result with NREL PHASE VI results for attached flow conditions [8]

J Laursen (2007) -: This paper presented the CFD analysis of Siemens SWT 23-93 whose blade length 45m. For CFD analysis they use ANSYS CFX 10.0 and 11.0 solvers. In solvers they use incompressible Reynolds Average Navier-Stokes equation (RANS) and SST turbulence model. Creates a computational domain for accomplish the analysis. Obtain a various results at various input parameters for modelled blades are as follows [9]:

Results	Wind Velocity and Radial position	Software result compared with measured result
Force Distribution	6m/s, 8m/s, 10m/s, 11m/s	CFX Transition Model, CFX Fully Turbulence Model, Ellipsys, Xblade
Axial Induction Factor	6m/s, 10m/s	CFX Transition Model, CFX Fully Turbulence Model, Xblade
Angle of attack, $C_{L,}$ C_D	6m/s, 10m/s	CFX Transition Model, CFX Fully Turbulence Model, Xblade
Pressure Coefficient and Skin Friction Coefficient	6m/s, r/R = 0.25, r/R = 0.50, r/R = 0.75	CFX Transition Model, CFX Fully Turbulence Model,

David Hartwanger* (2008) -: Author 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 k-epsilon turbulence model. There are two main aims for doing analysis are as:

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

Authors fulfill these two aims with one boundary condition which is pressure use as a inlet condition [10].

R.S. Amano (2009) -: This paper presented the optimization of aerodynamic design of wind turbine rotor blade by CFD with two case

a. Straight edge blade
b. Swept edge blade

They construct different domain shapes for accomplish the analysis. For analysis it uses different conditions in CFD solvers:

Wind Velocity	5m/s to 25m/s
Turbulence Model	k-ω SST
Reference Frame	Moving

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 [11].

N. S. Tachos (2009) -: Author investigated the aerodynamic analysis of Horizontal Axis Wind Turbine by using CFD codes of FLUENT. During CFD code they combine Reynolds Average Navier-Stokes equation with Spalart-Allmaras turbulence model and predict the span-wise loading on wind turbine. For analysis of HAWT they use NREL S809 aerofoil profile for wind turbine blade with a rotor radius of 5.029m and whose chord length is 0.4572m. It uses various boundary conditions during analysis are as follows [12]:

Wind Speed	7.2m/s
Rotational Speed	71.68 rpm
Reference Frame	Single
Pressure	PRESTO Scheme
Momentum	QUICK Scheme
Pressure Velocity Coupling	Simple

S. Gomez-Iradi (2009) -: Researcher done CFD analysis of HAWT with compressible Navier-Stokes equation at different conditions such as

a. Three wind velocity as inlet = 7m/s, 10m/s, 20m/s
b. Rotational speed or rotor = 72 rpm
c. Yaw angle = 0°

They construct a various domain size for doing analysis and use ICEM – HEXA for grid generation. It got better pressure distribution, torque, thrust and wind tunnel wall effect at 7m/s and 0° Yaw angle [13].

S. Rajakumar (2010) -: Here presented the numerical simulation of Horizontal Axis Wind turbine blade by CFD solver ANSYS FLUENT. For performing analysis it creates the 3D model of wind turbine rotor blade which is of NACA 4420 airfoil profile. In FLUENT for wind velocity 10m/s and angle of attack for 0° to 15 they got various results as follow:

a. It calculates lift and drag forces at various angle of attack
b. It obtains high lift and drag ratio at 5° angle of attack
c. Also calculate the coefficient of lift (C_L), coefficient of drag (C_D), and coefficient of pressure (C_P) at various angle of attack and obtain a good increase in C_L at 14° of angle of attack, C_P at 0° to 10°.
d. It also calculate the velocity distribution and pressure distribution at four different angle of attack are 0°, 5°, 10°, 15°.

All obtained results validate with their experimental results [14].

C. Rajendran (2011) -: This paper presented the CFD analysis of HAWT with incompressible Navier-Stokes equation. For analysis it take some initial conditions for obtained the pressure results are [15]:

Wind Velocity	12.5m/s
Yaw Angle	0°
Rotational Speed	25 rpm
Turbulence Model	k-ω SST
Reference Frame	Moving

Kamyar Mansour (2011) -: Author done CFD analysis Horizontal Axis Wind Turbine with three different turbulence models are

a. Standard k - ε
b. RNG k - ε
c. Spalart-Allmaras

And compare the result of these three turbulence model with experimental data of NREL. For analysis it creates a 3D model of NREL S809 airfoil series of wind turbine blade. It also creates the cylindrical domain and divided into two zones which are:

a. Moving Reference Frame
b. Stationary Reference Frame

They got pressure coefficient at two different wind speeds such as 7.2m/s, 10.5m/s which distributed at different span section of wind turbine blade. It takes rectangular grid for faces and hexahedral grid for 3D volume during analysis [16].

LU Qunfeng (2011) -: This study presented the power coefficient and pressure coefficient predicted at different tip speed ratio with k-ω SST turbulence model in FLUENT 6.3 version. In FLUENT it set little parameters which are:

Pressure based steady	Solver
Implicit	Solver
Momentum	Second order upwind scheme
Turbulent Kinetic Energy	Second order upwind scheme
Turbulent Dissipation Rate	Second order upwind scheme
Pressure Velocity Coupling	Coupled
Velocity of wind	7m/s
Pitch Angle	-2°
Rotational Speed	111.41 to 334.23 rpm
Tip Speed Ratio	2 to 6
Reynolds Number	$2.1*10^{-5}$

For analysis it construct the cylindrical domain whose radius 5 times the blade length [17].

Dong-Hyun Kim (2011) -: This study investigated the aero-elastic response and performance of 5MW wind turbine by using CFD and Computational Flexible Multi Body Dynamics (CFMBD). For solving the unsteady flow problem by using Reynolds Average Navier-Stokes equation which is combine with k-ω SST turbulence model. They also performed structural analysis with Finite Element Method and Fluid Structure Interaction. For analysis it creates the 3D model whose blade length= 62m, total diameter of rotor = 128m. It also creates the cylindrical domain for CFD analysis. For both CFD and CFMBD analysis it used FSIPRO 3D software. In FSIPRO 3D software used some parameters are [18]:

Wind velocity = 13m/s and 16 m/s

Rotational speed = 12 rpm

Moving reference Frame

Jun-Yong Lee (2011) -: Author developed a 1KW HAWT rotor blade for Korea where relatively low wind speeds. For modelled the blade it use NACA 63(2)-415 airfoil series whose blade radius R = 1250 mm. They predicted the lift coefficients (C_L) and drag coefficients (C_D) at different angle of attack whose range 4° to 9° for modelled blade with the help of XFOIL software. It also investigated the power coefficients, pressure distribution and velocity distribution for modeled blade with changes the tip speed ratio whose range taken 3 to 10 by using CFD solver ANSYS CFX 12. It obtains good lift to drag ratio when angle of attack is 7.5°. For analysis it created the cylindrical domain whose dimensions are [19]:

First half length = 8R

Second half length = 12R

Radius of cylinder = 5R

(Where R = Radius of blade)

In CFX it used parameters for analyzed the blade are:

Velocity = 6.32m/s

Rotational speed = 500 min^{-1}

k-ω SST turbulence model

Sugoi Gomez-Iradi -: Researcher done CFD analysis of wind turbine for investigate the blade loads and power of the wind turbine. For analysis it use Wind Multi Block (WMB) solver which is capable of solving the Unsteady Reynolds Average Navier-Stokes equation. In CFD it used ICEM v11for grid generation. Results of Local Flow Angle (LFA) and Span-wise Flow Angle (SFA) for isolated rotor case which is obtained by using Wind Multi Block (WMB) solver. The five different velocities condition used for obtained Local Flow Angle and Span-wise Flow Angle are, 5m/s, 7m/s, 10m/s, 13m/s, 20m/s.

It compared the results of 2D and 3D CFD analysis at incidence angle for four different cases which are:

a. Non Yawed flows in isolated rotor
b. Downwash effect

c. Yawed flow in isolated rotor
d. Full wind turbine

They compared the result of wind multi block (WMB) and results of 2D & 3D with experimental results of NREL [20].

Yuwei Li (2012) -: Researcher done the CFD analysis of NREL PHASE VI wind turbine rotor blade used dynamic overset method. For create the 3D model it use NREL S809 airfoil series where the dimensions of blade are:

Number of blades = 2
Rotor Diameter = 10.058m
Hub Height = 12.192m

They used Unsteady Reynolds Average Navier-Stokes equation (URANS) and Detached Eddy Simulation (DES) turbulence model in solver. It got pressure distribution, total power and thrust with various initial parameters such as:

Five different wind velocity as = 0m/s, 5m/s, 10m/s, 15m/s, 25m/s
Constant Pitch Angle = 3°
Rotational speed of rotor = 72 rpm

It also calculates the force using surface overset method and for area it used double-counting overset method. For whole analysis O type grid technology used [21].

Ji Yao (2012) –: Author creates the 2D model for the unsteady numerical simulation of VAWT by FLUENT software. For numerical simulation it used two turbulence models. They use NACA 0018 airfoil series for created the blade 2D model where the dimensions of blades are:

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	Second order upwind scheme
Velocity	Second order upwind scheme
Turbulent Kinetic Energy	Second order upwind scheme
Turbulent Dissipation Rate	Second order upwind scheme
Momentum	Second order upwind scheme
Pressure Velocity Coupling	SIMPLEC Algorithm
Grid Generation	ICEM Software
Turbulence Model	Standard k-ε, RNG k-ε,
Grid Type	Sliding Grid Technology

They investigated the velocity distribution, pressure distribution and change in torque for these two turbulences models. The effect of turbulence models on velocity distribution is lees but at pressure distribution is large and the torque from RNG k-ε model is larger than Standard k-ε model [22].

Huimin Wang (2012) -: This paper presented the numerical simulation of Vertical Axis Wind Turbine with Reynolds Average Navier-Stokes equations and Realizable k - ε turbulence model at different wind velocity. It used FLUENT software for performed CFD analysis. They use NACA 0018 airfoil series for created the blade 2D model where the dimensions of blades are:

Chord length of the blade **= 0.**1, Diameter of rotor = 0.9m, Rotational speed = 100 rpm

And also created the C-H type domain for CFD analysis whose dimension are:

C is a half semicircle shape whose radius = 16m
H is a rectangle whose size are = 32m * 30m

They obtained three different results at four different velocities are:

Results	Velocity
Contours of Velocity	10m/s, 15m/s, 20m/s, 25m/s
Distribution of Eddy	
Change of Torque	

Results shows wind velocities increases eddy existed in downstream region, total torque coefficients tend to smooth, velocity distribution at upstream is large [23].

CONCLUSION

The whole paper presented the computational fluid dynamics (CFD) analysis of wind turbine rotor blade where the 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. All simulation result calculation from CFD software plays a significant role for modeling the wind turbine rotor blade which is working efficiently or not the present given condition. The CFD analysis follows the basic theory and assumptions of Blade Element Momentum Theory (BEM).

REFERENCES

1. Dr. Gary L. Johnson Wind Energy Systems. Prentice-Hall. Manhattan, KS, January 1994.
2. Dr. Ibrahim Al-Bahadly Wind Turbines. Published by Intech, Janeza Trdine 9, 51000 Rijeka, Croatia, March 2011.
3. Mukund R. Patel Wind and Solar Power System. CRC Press LLC, New-York 1999.
4. Anderson J. D. Jr. Computational Fluid Dynamics. McGraw-Hill Inc.: New-York, 1995.
5. Ferziger J.H. and Peric M. Computational Methods for Fluid Dynamics. Springer-Verlag; Berlin, 1995.
6. C. A. J. Fletcher. Computational Techniques for Fluid Dynamics 1. Springer-Verlag; Berlin, 1988, 1991.
7. Nilay Sezer – Uzol and Lyle N. Long, ‘3- D Time– Accurate CFD Simulations of Wind Turbine Rotor Flow Fields’, AIAA Paper- No. 2006-0394.
8. E Ferrer and X Munduate, ‘Wind Turbine Blade Tip Comparison Using CFD’, Journal of Physics: Conference Series 75 (2007) 012005, doi: 10.1088/1742-6596/75/1/012005.
9. 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, doi: 10.1088/1742-6596/75/1/012007.
10. David Hartwanger and Dr. Andrej Howat, ‘3D Modeling of a Wind Turbine Using CFD’, NAFEMS Conference, 2008.
11. 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.
12. N.S. Tachos, A.E. Filios, D.P. Margaris and J.K. Kaldellis, ‘A Computational Aerodynamics Simulation of the NREL Phase II Rotor’, The Open Mechanical Engineering Journal, 2009, Volume 3, 9-16.
13. S. Gomez- Iradi, R. Steijl, G.N. Barakos, ‘Development and Validation of a CFD Technique for the Aerodynamic Analysis of HAWT’, Journal of Solar Energy Engineering, August 2009, Vol. 131/ 031009-1.
14. S. RajaKumar, Dr. D. Ravindran, ‘Computational Fluid Dynamics of Wind Turbine Blade at Various Angle of Attack and Low Reynolds Number’, IJEST, Vol. 2(11), 2010, 6474-6484.
15. 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.
16. Kamyar Mansour, Mohsen Yahyazade, ‘Effects of Turbulence Model in Computational Fluid Dynamics of HAWT Aerodynamic’, WSEAS Transactions on Applied and Theoretical Mechanics, Vol. 6, July 2011.
17. Lu Qunfeng, CHEN Jin, CHENG Jiangtao, QIN Ning, Louis Angelo M. Danao, ‘Study of CFD Simulation of a 3D Wind Turbine’ IEEE 2011.

18. Dong-Hyun Kim and Yoo-Han Kim, ‘Performance Prediction of a 5 MW Wind Turbine Blade Considering Aero-elastic Effect’ World Academy of Science Engineering and Technology, 81 2011.
19. Jun-Yong Lee, Nak-Joon Choi, Jong-Won Lee, Han-Yong Yoon and Young-Do Choi, ‘Shape design and CFD Analysis on a 1KW Class HAWT Blade for Hybrid Power Generation System’ The 11_{th} Asian International Conference on Fluid Machinery and The 3_{rd} Fluid Power Technology Exhibition, Paper ID: AICFM_TM_027 November 21-23, 2011, IIT Madras, Chennai, India.
20. Sugoi Gomez-Iradi, George N. Barakos, Xabier Munduate, ‘A CFD Investigation of the Near- Blade 3D Flow for a Complete Wind Turbine Configuration’.
21. Yuwei Li, Kwang-Jun Paik, Tao Xing, Pablo M. Carrica, ‘Dynamic Overset CFD Simulations of Wind Turbine Aerodynamics’, ELSEVIER, Renewable Energy 37 (2012) 285-298.
22. 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.
23. Huimin Wang, Jianliang Wang, Ji Yao, Weibin Yuan, Liang Cao, ‘Analysis on the Aerodynamic Performance of VAWT Subjected to the Change of Wind Velocity,’ ELSEVIER, International Conference on Advances in Computational Modeling and Simulation, Procedia Engineering 31 (2012) 213-219.