International Journal of Engineering and Applied Sciences (IJEAS)
ISSN: 2394-3661, Volume-3, Issue-4, April 2016

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

Ms. Priti G. Bhadake, Prof. V. G. Gore

***Abstract*— Energy requirement is increasing day to day and therefore renewable energy source is most power full way to full fill requirement of energy. Wind energy is one of the most efficient renewable energy. The aerodynamic airfoils of wind turbine blades have crucial influence on aerodynamic efficiency of wind turbine. This involves the selection of a suitable airfoil section for the proposed wind turbine blade. Airfoil shape of wind turbine and design of airfoil shape are investigated in this review paper. General airfoil behavior and airfoil optimization for improve coefficient of lift are to be investigate in this review paper. The aerodynamic airfoils of wind turbine blades have crucial influence on aerodynamic efficiency of wind turbine. This involves the selection of a suitable airfoil section for the proposed wind turbine blade. Airfoil is most affecting parameter in whole wind turbine for extracting energy so it is necessary to design for maximizes aerodynamic efficiency for high energy production rate.**

***Index Terms*— Computational Fluid Dynamics (CFD), NACA**

I. INTRODUCTION

A wind turbine is a rotary device that extracts energy from the wind. 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. Rotor blade is a key element in a wind turbine generator system to convert wind energy in to mechanical energy. Most blades available for commercial grade wind turbines incorporate airfoil shaped cross sections. These blades are found to be very efficient at lower wind speeds in comparison to the potential energy that can be extracted. Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary condition.

There are several technique is reviewed for design an airfoil and optimization of airfoil shape for maximum coefficient of lift force. And review CFD (Computational Fluid Dynamics) analysis which is very highly using for prediction of aerodynamic behavior of airfoil.

II. LITERATURE REVIEW

SrinivasG et al (2014) has suggested that blades play an important role in converting wind energy into electrical energy. When the Wind attack to the blade, reaction force produce in form of lift and drag forces , this Forces are Calculated by Numerical method but because of complication and also it is a tedious work we can also go for another method , That is Computation Fluid Dynamics (CFD) method for getting desired results. CFD method is based on fluid mechanics, Its Function is to use blade air-profile and consider a 2 dimensional profile Choose by Design Foil Workshop for various chords (Abbott et al Report No.824) at different Angle of Attack of air and also in Different Reynolds Number.

R. B. Gowardhanand et al (2014) has presented an aerodynamic design of blade using CFD analysis. The objective of this project is to increase the efficiency of wind turbine by reducing the drag and lift. This present work is done in designing a wind turbine blade by using CREO-ELEMENT/PRO 5.0 Software and optimizing the blade aerodynamically by using CFD analysis in ANSYS 11.0 Software. In this project a wind turbine blade is design by using CREO-ELEMENT/PRO 5.0 Software. And optimize the blade aerodynamically by using CFD analysis in ANSYS 11.0 Software. The maximum value of coefficient of performance (CPmax = 0.44) was observed at velocity of air 3 m/s. and 6 rpm. This blade can generate maximum power of 64199.73 Watts at maximum CP. From analysis we concluded that ANSYS-Fluent shows a good performance in calculating the lift, drag and moment coefficients of aerofoils. The wind turbine blade designed by this method has good aerodynamic performance in low wind speed conditions.

Mayurkumar kevadiya et al. (2013) has study the aerodynamic airfoils of wind turbine blades have crucial influence on aerodynamic efficiency of wind turbine. This involves the selection of a suitable airfoil section for the proposed wind turbine blade. In this paper NACA 4412 airfoil profile is considered for analysis of wind turbine blade. Geometry of the airfoil is created using GAMBIT 2.4.6. And CFD analysis is carried out using FLUENT 6.3.26 at various angles of attack from 0° to 12. The coefficient of lift and drag values are calculated for 1 ×105 Reynolds number. And it result the coefficient of Lift and drag is calculated for this NACA 4412 series for the angle of attack 0° to 12°. The coefficient of Lift/Drag ratio increases with increase in Angle of attack up to 8°. After 8°, Lift/Drag ratio decreases with increase in Angle of attack.

HardikPatel et al (2013) has carried out Computational Fluid Dynamics (CFD) analysis of wind turbine blade with complete drawing and details of sub-system. First the type of airfoil is used is decided. Then find out the airfoil co-ordinate for drawing the airfoil shape. Here NACA 0018 selected for the analysis. Here the maximum thickness is 18% which indicate the maximum thickness (in per cent of the chord).In this analysis, the geometry is prepared in the pro-e software package and then after it is saved in iges format. Then import this geometry in the ANSYS 12.0.

Ms. Priti G. Bhadake, Scholar M.E. (2nd year/ Thermal Engg.), P. R. Pote Engg. Colg. Amravati, (M.S.), India
Prof. V. G. Gore, Assistant Professor, P. R. Pote Engg. Colg. Amravati, (M.S.), India

Form this study they conclude as following:

(1)The maximum value of coefficient of performance (CPmax = 0.277191) was observed at angle of attack 60 and 70 and the velocity of 26 m/s.

(2) This blade can generate maximum power of 3374 w at maximum CP, at angle of attack 60 and velocity of air 26m/s.

(3) It was observed that value of numerical power increases as angle of attack increases from 10 to 70, after 70 the value of numerical power reduced. Hence critical angle of attack for this blade is 70.

Fei-Bin Hsiao et al (2013) has suggested three different HAWT use at a different condition like Optimum Blade Shape, Optimum twist blade and Untwist blade, the HAWT blade geometry is a NACA 4418. The above this condition is to be experimented in CFD software use a k-ω SST turbulence model. We get some result in experiment work, that the Optimum Blade (OPT) is more efficient to the untapped and optimum twist blade at a wide range of power coefficient of tip speed ratio which from 4.5 to 7 and untapered and untwist blade operates in lowest Cp value..

Farooq Ahmad Najar et al. (2013) in presented paper they done CFD analysis of NREL S809 Airfoil by selecting various numbers of solver and compare it to experimental data of wind tunnel test and conclude that k-e standard wall function is best match with experimental data than other solver like a S-A, SST, etc. NREL S809 gives a maximum performance at 140, angle of attack.

R. Mukesh et al. (2013) in presented paper they have selected a NACA 2411 airfoil as base shape for optimization process. The airfoil was described using the PARSEC parameterization scheme. The flow around the airfoil was solved using the Panel method. And finally optimized airfoil shape is validating by using an experimental validation. GA is used for optimization and at 5 of Angle of Attack airfoil analysis is carried out.

Monir Chandrala et al. (2013) in this paper, author selected 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 different. 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° for 32 m/s wind speed. In this paper flat blade with single airfoil is considered for an analysis. Fig. 2.6 for CFD analysis of blade at 10° angle is showing here.

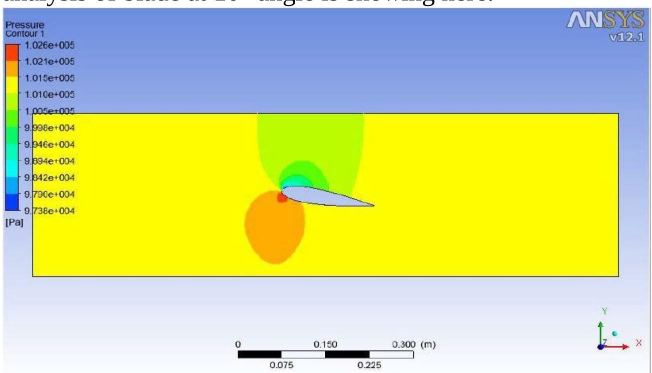

Fig. 2.6: Pressure Contour on Blade at 10° Angle

H. V. Mahawadiwar et al. (2012) in this research paper, author have an already a design of blade, which is use for CFD analysis. They modeling that in the Pro-E and then generate a mesh in GAMBIT and after then CFD analysis is carried out in fluent software. They also compute Numerical power and conclude that the numerical power is increase as AoA increase from 0° to 7°, after 7° the value of Numerical power is suddenly reduce from these effect we can say that after 7° AoA there is a stall effect is produce. Maximum value of Co-efficient of power was observed at AoA 7° and velocity of wind 8m/s. They also give a maximum power generation by blade 620W ate maximum co-efficient of power, and AoA 7° at velocity of wind 8m/s.

J. Fazil et al. (2011) in this paper, author proposed a quantic reverse engineering Bezier curve formula for producing airfoil shape. By, this formula they produce an airfoil shape in CATIA and validate with NACA four digit profile generator. They used a quantic reverse engineering Bezier curve formula for the find out the control points of the camber profile which is used to create upper and lower camber profile .By using the control points, we easily modify the shape of the profile so that to produce the cambered airfoil shape without affecting basic airfoil geometry. The objective of this work is to find a simple and accurate way to design the airfoil profile in CATIA using six camber control point position. However the proposed method is applied only for six camber control point position in the airfoil. Below Fig. 2.2 show a validation of profile using a CATIA and Fig. 2.3 show a modification using a control point in CATIA software.

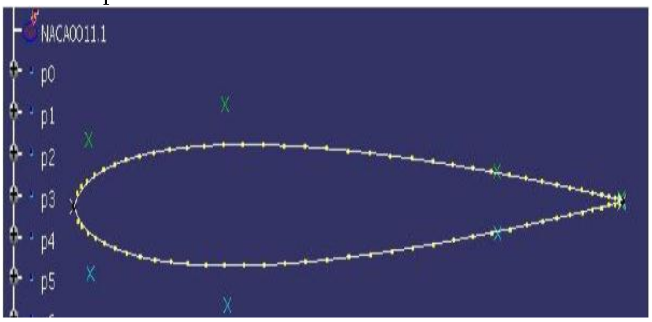

Fig. 2.2: Validation of NACA Profile Generator with CATIA Profile

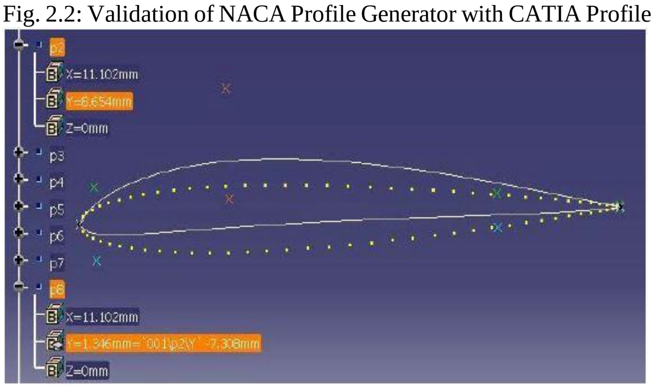

Fig. 2.3: Modification of Airfoil Profile by Control Point in CATIA

C. Rajendran 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 k-ω SST.

Chalothorn Thumthae et al. (2009) in this paper, author have experimental data of profile NREL's S809 and work on finding a optimal angle of attack for untwisted blade by use of CFD analysis for prediction of AoA. They analyze that the angles are slightly larger as the speeds are higher and this is consistent with the shift of the curves as the Reynolds numbers are increased.

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 k- 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.

E Ferrer et al (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 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. 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.

Dr. S. P. Vendan et al carried out in his paper a Horizontal axis wind turbine blade profile NACA 63-415 is analyzed for various angles of attack. The coefficient of Lift and drag is calculated for this NACA 63-415 for various angles of attack from 0° to 16° and the maximum ratio is achieved at 2° of angle of attack. The coefficient of Lift increases with increase in angle of attack up to 8°. After 8°, the coefficient of lift decreases and stall begins to occur. The drag force begin of dominate beyond this angle of attack. The rate of increase in lift is more for angle of attack from 0° to 8° and then it starts to decrease. The drag increase gradually until 5° angle of attack and then rapidly increases. The CFD analysis is carried out using STARCCM+ software. These results are compared with the wind tunnel experimental values for validation.

III. CONCLUSION

From the above reviews we can conclude that a number of researchers are using CFD to study wind turbine aerodynamic analysis of blade. There are several technique is reviewed for design an airfoil and optimization of airfoil shape for maximum coefficient of lift force. And review CFD (Computational Fluid Dynamics) analysis which is very highly using for prediction of aerodynamic behavior of airfoil.

REFERENCES

[1] Srinivas G , Mahesha G T, Chethan K N and Arjun N, 2014, Analysis of Wind Turbine Blade, International Journal of Current Engineering and Technology E-ISSN 2277-4106,P-ISSN 2347-5161.

[2] R. B. Gowardhanand, Mr. Ritesh k. Bais (2014) IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684, p-ISSN: 2320-334X PP 45-47.

[3] Mayurkumar kevadiya, 2D Analysis of NACA 4412 AIRFOIL International Journal of Innovative Research in Science, Engineering & Technology Vol. 2, Issue 5, May 2013.

[4] HardikPatel1, Sanat Damania, Performance Prediction of Horizontal Axis Wind Turbine Blade, International Journal of Innovative Research in Science, Engineering & Technology, Vol. 2, Issue 5, May 2013.

[5] Fei-Bin Hsiao, Chi-JengBai and Wen-Tong Chong, The Performance Test of Three Different Horizontal Axis Wind Turbine (HAWT) Blade Shapes Using Experimental and Numerical Methods, www.mdpi.com/journal/energies, Energies 2013, 6, 2784-2803; doi:10.3390/en6062784.

[6] Farooq Ahmad Najar, G A Harmain, Blade Design & Performance Analysis of Wind Turbine, International Journal of Chem Tech Research, Vol.5, No.2, pp 1054-1061, April-June 2013.

[7] R. Mukesh, K. Lingadurai, U. Selvakumar, Airfoil shape optimization using non-traditional optimization technique and its validation, Journal of King Saud University – Engineering Sciences (2014) 26, 191–197, http://dx.doi.org/10.1016/j.jksues. 2013.04.003.

[8] Monir Chandrala, Abhishek Choubey, Bharat Gupta, CFD analysis of horizontal axis wind turbine blade for optimum value of power, International Journal Of Energy And Environment, Volume 4, Issue 5, Pg. No. 825-83, 2013.

[9] 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 (IJERA), Vol. 2, Issue 3, Pg. No. 3188-3194, 2012.

[10] J. Fazil and V. Jayakumar, Investigation of Airfoil Profile Design Using Reverse Engineering Bezier Curve, Asian Research Publishing Network (ARPN), VOL. 6, No. 7, 2011.

[11] 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.

[12] Chalothorn Thumthae, Tawit Chitsomboon "Optimal angle of attack for untwisted blade wind turbine", Renewable Energy 34, Pg. No. 1279–1284, 2009.

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

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

[15] Dr. S. P. Vendan, S. Aravind Lovelin, M. Manibharathi and C. Rajkumar, Analysis of a Wind Turbine Blade Profile for Tapping Wind Power at the Regions of Low Wind Speed, Vol-2, Issue-2