A Review of optimization of Airfoil Shape Used in Wind Turbine Blade Application

Abhishek Prasad[1], Vikky Kumhar[2], Ajay Kumar Verma[3]
[123]Department of Mechanical Engineering, SSTC-SSGI, Bhilai (C.G.) India.

Abstract

Computational fluid dynamics (CFD) is one of the computer-based solution methods which is more widely employed in aerospace engineering. The computational power and time required to carry out the analysis increase as the fidelity of the analysis increases. Aerodynamic airfoil curve shape optimization has become a vital part of aircraft design in the recent years. The objective of this work is to introduce the knowledge of describing general airfoil using flow parameters by representing its curve shape.

***Keywords**: Wind Turbine Blade, Airfoil, CFD, Optimization.*

I. Introduction

Wind energy is one of the most promising renewable energy sources under development. Comparing other renewables wind power generation is the fastest growing industry in energy sector. By the end of 2012, the world total installed capacity of wind turbines reached 285 GW and it is predicted that will exceed 500 GW by the year 2017. For the past years, it was common practice to use existing airfoil families, like the well-known S and NACA series, for the design of wind turbine blades, however the need of furthering wind turbine technologies has led to the quest for alternatives.

1.1 Airfoil Characteristics on Rotor Performance

Depending on the size of the blades, the airfoil characteristics of the wind turbine defines the aerodynamic performance, amount of mechanical power of the rotor, and rigidity of the blade. It is important to understand the aerodynamic concepts of the airfoils to define the power production of the given wind turbine.

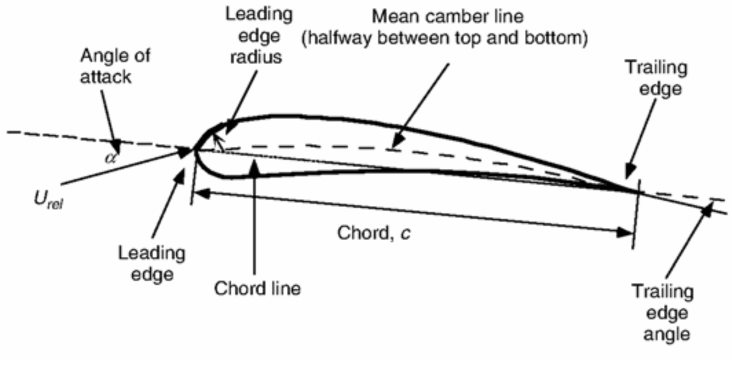

Figure I.1 Sketches of the typical airfoil

II. Literature Review

The literature review presented in the work are classified into three major categories namely Wind turbine airfoil aerodynamics, Design and performance of wind turbine, Optimization of wind turbine and simulation techniques.

Fuglsang and Madsen (1999), a numerical multi-disciplinary optimization method for design of horizontal axis wind turbines was proposed. The objective was to minimize the cost of generating energy. They considered design fatigue load, extreme wind loads and annual generation of energy. They developed an empirical approach to identify the sensitivities of the above parameters. The empirical approach saved substantial computing time. They considered 1.5 MW stall regulated rotor for optimization. They optimized the shape of the rotor to withstand the maximum strain and for economical use of material.

Hansen (2008) when airflow with no disruption and smooth streamline is called laminar flow. The particles of the air move parallel layer. In turbulent flow, the fluctuations cause the particles of the air to get mixed and move through the layers; as a result, they get high and low kinetic energy, giving a much more uniform velocity profile. Also, turbulent flow has higher friction and lower tendency to separate.

Arvind (2010) researched on NACA 4412 airfoil and analysed its profile for consideration of an airplane wing. The NACA 4412 airfoil was created using CATIA V5 and analysis was carried out using commercial code ANSYS 13.0 FLUENT at a speed of 340.29 m/sec for angles of attack of 0˚, 6, 12 and 16˚. k-ε turbulence model was assumed for Airflow. Fluctuations of static pressure and dynamic pressure are plotted in form of filled contour.

kevadiya (2013) studied the NACA 4412 airfoil profile and recognized its importance for investigation of wind turbine edge. Geometry of the airfoil is made utilizing GAMBIT 2.4.6. Also, CFD investigation is done utilizing FLUENT 6.3.26 at different approaches from 0˚ to 12˚.

Saxena and Agrawal (2014) analyzed a basic aerodynamic theory of wings and the provided and methodology for wind tunnel testing. They found angle of attack at which the lift is maximized in order to get the best performance of this wing when in flight.

Li et al., (2015) employed a detailed investigation of the aerodynamic characteristics of airfoil RAE2822 at different ground clearances and AOAs is carried out. It shows: Realizable k-epsilon model is considerably demonstrated with good capability of predicting the flow characteristics. In study of the ground effect of RAE2822, high lift and ratio of lift and drag only could be achieved in medium angle of attack. The stationary ground condition would induce the separation of boundary layer on the ground and influence the aerodynamic characteristic when the airfoil is close to the ground. The compressibility of air shows a significant impact on the aerodynamic characteristics of airfoil in ground effect, it should be considered in numerical simulation and experiment.

Patil and Thakre (2015) an attempt is made to investigate the Lift and Drag forces for different Reynolds number and angle of attack for wind turbine blade. In present work NACA 0012 airfoil profile is considered for analysis of wind turbine bladeThe Lift and Drag forces are calculated at different angle of attack varying from 0oto 80ofor Reynolds number from 10,000 to 800000 by Computational Fluid Dynamics (CFD) analysis. The validations of the present work are done by comparing the results obtained from analysis with experimental results obtained by Sandia National Laboratories (SNL) energy report.

Matyushenko et al., (2017) work in flows around sets of airfoils with different shapes and thicknesses have been numerically investigated at relatively high Reynolds numbers ($Re \geq 10^6$) and low turbulence intensity ($I \leq 0.1\%$) using two-dimensional (2D) Reynolds-Averaged Navier-Stokes equations (RANS) closed by different turbulence models. The effects of a set of factors such as wind tunnel walls, the compressibility and the influence of the laminar-turbulent transition were investigated. The most likely reasons for the systematic disagreement between simulation and experimental data were established to be 3D effects ignored by 2D simulation, and imperfection of the existing turbulence models

Geng et al., (2018) numerically reproducing wind tunnel experimental tests of flow behavior around a pitching airfoil is a challenge, especially under the occurrence of dynamic stall at relatively high angles of attack. This not only requires the application of advanced turbulence models, but also asks for examining the influence of the most relevant model parameters in detail. This contribution presents the results of an extensive computational study of the Unsteady Reynolds-Averaged Navier-Stokes (URANS) simulations whereby the flow characteristics are simulated flow around a pitching NACA0012 airfoil at a Reynolds number of 1.35 x 10^5, as obtained by first performing 2D by the Transition SST turbulence model.

Mouhsine et al., (2018) worked in Aerodynamics and structural analysis of wind turbine blade. The ultimate objective is to increase the reliability of wind turbine blades through the development of the airfoil structure, to calculate an optimum blade shape for the procedure begins with the choice of airfoils characteristics. Then an initial wind blade design is determined using blade element momentum. The blade plays a pivotal role, because it is the most important part of the energy absorption system.

Anitha et al., (2018) The efficient aerodynamic performance is carried out by selection of wing airfoil. The optimal shape of the airfoil generates high lift and low drag within the design constraints often imposed by the structural requirements. In the present study, the optimization of airfoil shape is carried out using genetic algorithm, particle swarm optimization methods for a typical airfoil NACA4412 in MATLAB environment. Airfoil parameterization is done using cubic spline, PARSEC and CST methods.

III. Conclusion

The various airfoils are experimentally tested as well as computational fluid dynamics and the results are presented by many researchers in the literature. There are no generalized correlations that could be applied to predict the coefficient of lift and drag for various NACA and S types airfoils in wind turbine blade application. The main design principles and failure mechanisms of blades in operation are assessed and explained from an industry point of view, in a realistic manner.

The effect of Reynolds number on coefficient of lift and drag has been attempted by some of the researchers using experimental set-up. In the present study, an iterative approach for computing the performance of the wind turbine airfoil is developed to compute the axial and tangential flow factors.

Dependent on whether the goal is to improve on an existing design or to create a completely new design, different methods are often required. If the new design only requires small changes to the initial geometry. The efficient aerodynamic performance is carried out by selection of wing airfoil. The optimal shape of the airfoil generates high lift and low drag and high thermal performance within the design constraints often imposed by the structural and thermal requirements.

References

[1] P. Fuglsang and H. A. Madsen, “Optimization method for wind turbine rotors,” J. Wind Eng. Ind. Aerodyn., vol. 80, no. 1–2, pp. 191–206, 1999.
[2] Hansen, M. O. L., “Aerodynamics of wind turbines. London: Earthscan, 2008.
[3] Arvind M. “CFD Analysis of Static Pressure and Dynamic Pressure for NACA 4412” International Journal of Engineering Trends and Technology, Volume 4, Issue 8, pp 3258-3265, 2010.
[4] Kevadiya M., “CFD Analysis of Pressure Coefficient for NACA 4412” International Journal of Engineering Trends and Technology, Volume 4, Issue 5, pp 2041-2043, 2013.
[5] G. Saxena and M. Agrawal, 2013, "Aerodynamic analysis of NACA 4412 airfoil using CFD", International Journal of Emerging Trends in Engineering and Development,4, pp.416-423.
[6] S. Li, D. Zhou, Y. Zhang, and Q. Qu, Aerodynamic Investigation of RAE2822 Airfoil in Ground Effect, vol. 126. Elsevier B.V., 2015.

[7] B. S. Patil and H. R. Thakare, “Computational Fluid Dynamics Analysis of Wind Turbine Blade at Various Angles of Attack and Different Reynolds Number,” Procedia Eng., vol. 127, pp. 1363–1369, 2015.

[8] A. Matyushenko, E. V. Kotov, and A. V. Garbaruk, “Calculations of flow around airfoils using two-dimensional RANS: an analysis of the reduction in accuracy,” St. Petersbg. Polytech. Univ. J. Phys. Math., vol. 3, no. 1, pp. 15–21, 2017.

[9] F. Geng, I. Kalkman, A. S. J. Suiker, and B. Blocken, “Sensitivity analysis of airfoil aerodynamics during pitching motion at a Reynolds number of 1.35×105,” J. Wind Eng. Ind. Aerodyn., vol. 183, no. March, pp. 315–332, 2018.

[10] S. El Mouhsine, K. Oukassou, M. M. Ichenial, B. Kharbouch, and A. Hajraoui, “Aerodynamics and structural analysis of wind turbine blade,” Procedia Manuf., vol. 22, pp. 747–756, 2018.

[11] D. Anitha, G. K. Shamili, P. Ravi Kumar, and R. Sabari Vihar, “Air foil Shape Optimization Using CFD and Parametrization Methods,” Mater. Today Proc., vol. 5, no. 2, pp. 5364–5373, 2018.