The 2013 World Congress on
Advances in Structural Engineering and Mechanics (ASEM13)
Jeju, Korea, September 8-12, 2013

Aerodynamic Design of 2.5 MW Horizontal Wind Turbine Blade in Combination with CFD Analysis

Seul-Ki Yeom*, Tae-Jin Kang*, Warn-Gyu Park‡

1) *School of Mechanical Engineering, Pusan National University, Busan 609-735, Korea*

Abstract

Recently, 2.5 MW offshore wind turbine has been used for commercial applications. Aerodynamic design and analysis of the wind turbine is necessary to reduce unnecessary processes. Thus, the aerodynamic design of a 2.5 MW offshore wind turbine blade was studied to improve its power performance. The blade is a very important part in the evaluation of the overall performance of the wind turbine system. The tangential induction factor, axial induction factor, chord length, and twist angle were calculated on the basis of BEM (blade element momentum) theory to predict the performance of the blade through numerical analysis. Among a variety of theories and approaches present to help create the most appropriate blade, BEM theory, which is relatively easy to apply and has high–accuracy has been applied the NACA64-618 and DU series airfoils of the designed blade. These airfoils were selected because their experimental values are easy to obtain, and it have been recognized by the National Renewable Energy Laboratory (NREL). The overall system is a model of a three-bladed horizontal axis wind turbine. In this study, we applied the blade aerodynamic design to 2.5 MW wind turbine and the result shows power, streamline around blade

Key words: Blade element momentum theory, Computational fluid dynamics, Wind turbine blade, Multiple reference frame

1. Introduction

Wind has become useful source of renewable energy in Korea, and the demand for wind power has been increasing daily. Therefore, the requirement for larger wind turbine has increased. In general, wind turbines are divided into two types—horizontal and vertical—according to the rotation direction. The horizontal axis wind turbine has greater possibility for blade pitch control over a wide range, and exhibits excellent output characteristics. Moreover, it requires a smaller installation space than the vertical type. wind turbine is expanding from onshore to offshore locations to achieve larger systems and higher efficiency. Offshore wind turbine presents a suitable alternative to the problems faced with onshore wind problems. Especially, high tower placement has less influence on the turbulence flow. Wind turbine blade

‡ Professor, corresponding author, *E-mail*: wgpark@pusan.ac.kr
* Graduate Student

is the basic elements of a wind turbine, and their optimized design can help improve the total system's performance.

In this study, a 2.5 MW horizontal axis wind turbine blade was designed and its aerodynamic performance was validated by CFD (computational fluid dynamic) analysis. The airfoils used in this study belong to the NREL 5 MW wind turbine system, because much experimental data are available on this system. Finally, this research verified the flow field and the aerodynamic performance to achieve the targeted output.

2. Blade Design of Wind Turbine

2.1 Design Geometry

For designing a wind turbine blade, first, the appropriate airfoil structures should be selected. The DU series and NACA64-618 airfoils used in the NREL 5 MW wind turbine blade system were considered as the basic design of the blade, because they are widely used and it has been validated that their lift-to-drag ratio is excellent at any position of the blade radius in large wind turbine systems. The range of change of airfoil thickness for each airfoil was from 18% to 40%. The wind turbine system parameters are listed in Table 1.

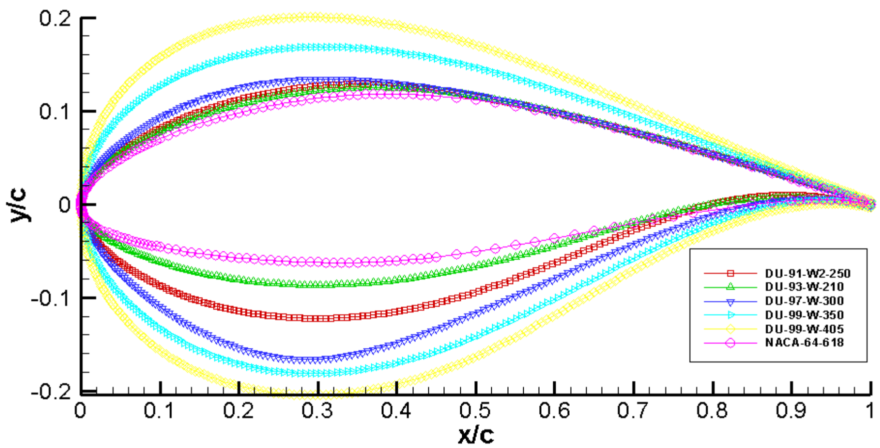

Fig. 1. Airfoil Shape of Wind Turbine Blade

2.2 Blade Design based on Blade Element Momentum Theory

In this study, the mathematical model implemented for the fluid dynamics design of a wind turbine blade is based on BEM theory. The BEM analysis combines momentum theory and blade element theory. Momentum theory refers to the analysis of blade forces based on conservation of linear and angular momentum, while blade element theory refers to the analysis of forces in concentric ring elements of rotor. With BEM theory, it is possible to analyze the change in the airfoil cross section based on the lift and drag at the position of the blade local. It is predicted accurately to analyze its performance. Here, the blade is divided of their rotational direction to obtain the optimized blade geometry. When the rotor blades operate in the post-stall region, BEM theory underestimates the evaluated rotor power as compared to the experimental results.

As shown in Fig. 3, since the rotational speed increases from root to tip, the design of each airfoil located at a different position along the blade span is optimized to achieve the optimal angle of attack according to the rated wind speed so that the twist angle is reduced. Where, r is local radius, R is radius of blade and c is chord length.

Table. 1. Main Design Parameters

Rated power	2.5MW
Design wind velocity	10m/s
Cut-in wind velocity	3m/s
Cut-out wind velocity	25m/s
Blade radius	47.096m
Blade number	3
Rated rotational speed	12.17rpm
Tip speed ratio	6
Rotor orientation	Upwind

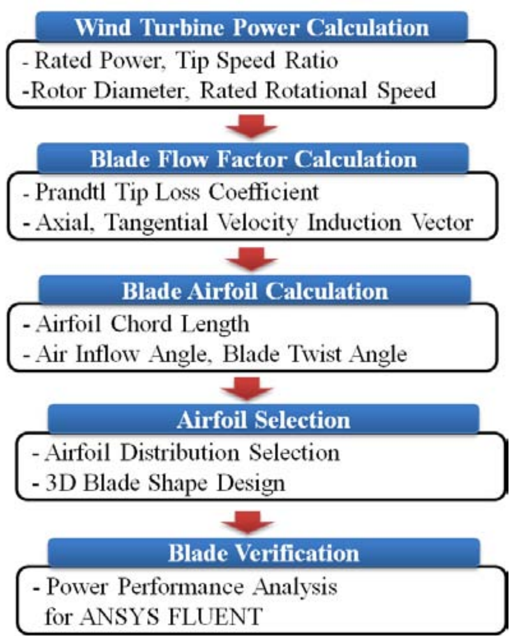

Fig. 2. Design Flow Chart of Wind Turbine Blade

3. CFD Analysis

3.1 Grid System and Boundary Condition

CFD modeling of the wind turbine blade was performed using ANSYS FLUENT V 13.0 in combination with the ANSYS ICEM CFD meshing tool. The geometry was determined using detailed specifications of the local section profile, chord length, and twist angle. The inlet section of domain is three times and the outlet section is five times based on radius R.

Table. 2. Distributed Blade Aerodynamic Properties

r/R	Radius[m]	Chord length[m]	Twist angle[deg.]	Airfoil
0	0	2.50	25.00	Cylinder
0.05	2.37	2.50	25.00	DU40_A17
0.10	4.75	3.44	25.00	DU40_A17
0.20	9.50	5.13	23.85	DU40_A17
0.30	14.21	3.67	17.44	DU35_A17
0.40	18.99	3.07	13.75	DU30_A17
0.50	23.74	2.70	11.34	DU25_A17
0.60	28.49	2.45	9.64	DU25_A17
0.66	30.86	2.37	8.99	DU25_A17
0.71	33.24	2.29	8.36	DU21_A17
0.81	37.99	2.14	7.31	NACA64_618
0.91	42.73	1.91	6.27	NACA64_618
1.00	47.10	1.50	5.24	NACA64_618

The overall grid system were formed that consist of approximately 4,000,000 nodes and feature hexahedral grid. The grid around the blade was concentrated to sufficiently express the surface pressure of the blade and fluid flow: the O-grid was used. Moreover, to ensure mesh quality at the trailing edge piled layer forming around the blade.

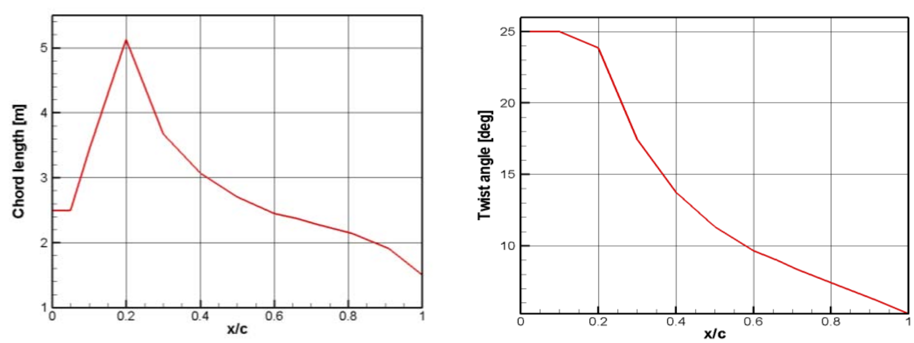

Fig. 3. Chord Length and Twist Angle v/s x/c

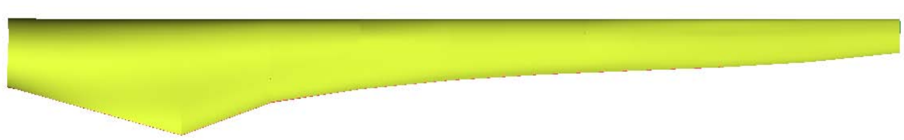

Fig. 4. Configuration of Wind Turbine Blade

The velocity boundary condition at the entrance was set as constant velocity and the pressure boundary at the outlet condition was set as constant pressure. At this point, the turbulence intensity set up was 0.03%. When the grid system has a large number of nodes, it takes a long computational time to simulate the stable flow around the blade and interface areas. The common rotor system analysis methods are SRF (single reference frame), MRF (multiple reference frame), MPM (mixing plane model), SMM (sliding mesh model), and DMM (dynamic mesh model). In this study, the MRF method was adopted. This method utilizes a different coordinate system considering the rotation of the blade; thus, this method has the advantage of a short analysis time.

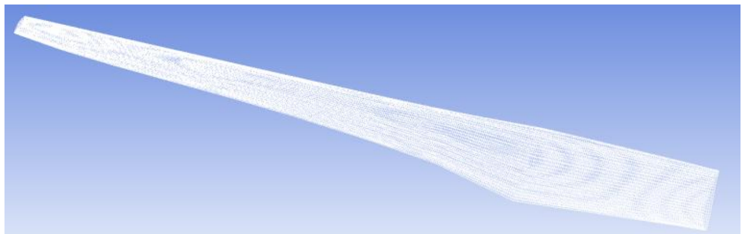

Fig. 5. Structured Mesh of Blade Surface

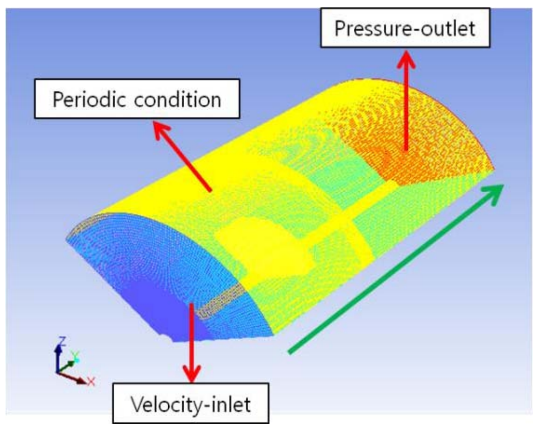

Fig. 6. Computational Domain and Boundary Condition

Table 3 lists the analysis conditions in this study and the obtained result for the corresponding tip speed ratio. The inflow velocity was set as 5 m/s, and the rotational speed was varied in the range from 3 to 12.17 rpm. The output value was observed, and the flow was controlled at 1 rpm near the rated speed value. The tip speed ratio was obtained using Eq.1.

$$\lambda = \frac{\omega \cdot R}{V_\infty}$$

(1)

where R is the blade radius, ω is the angular velocity, and V_∞ is the freestream velocity.

Table. 3. Conditions of Wind Turbine Blade Design Analysis

	V_∞[m/s]	Rotational Speed[rpm]	Tip Speed Ratio(TSR)
Case1	5	3	2.96

Case2	5	5	4.93
Case3	5	7	6.90
Case4	5	9	8.88
Case5	5	10	9.86
Case6	5	11	10.85
Case7	5	12.17	12.00

3.2 Numerical Method

The turbulence model was applied to the Spalart–Allmaras model [2]. The Spalart–Allmaras model is used for analysis when the separated flow is opposite to the freestream direction. In this analysis, the rate of convergence is higher as compared to the two-equation model and hence, it can be expected that the computation time can be reduced using this method.

4. Analysis and Discussions

In Fig. 7, the streamlines from the blade root to the tip show the distribution. The thickness of this zone is phenomenal because it is more than it is in other areas. At the blade tip, a complicated flow occurs, and when the distribution is viewed as a whole, it causes a reduction in the aerodynamic performance.

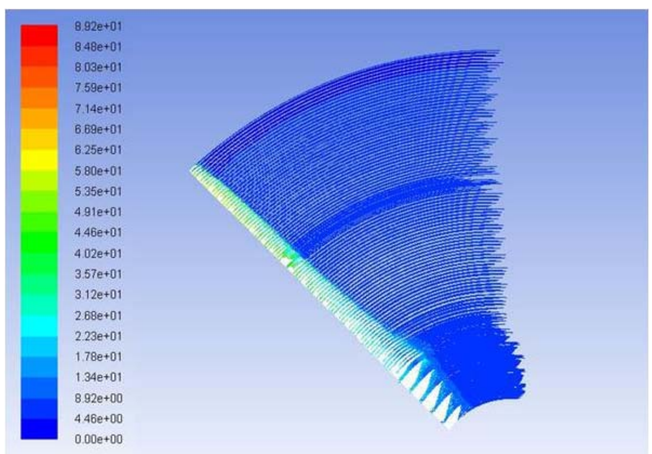

Fig. 7. Surface Streamlines at Rotational Speed of 12.17 rpm

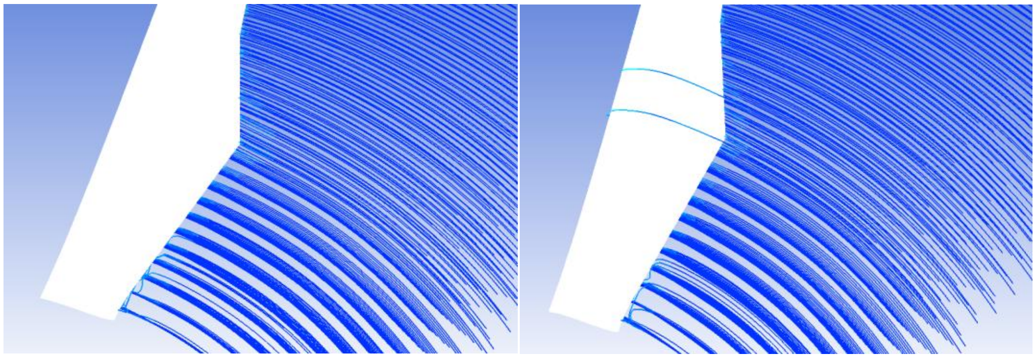
(a) 9 rpm (b) 12.17 rpm

Fig. 8. Streamlines of Pressure Distribution at Blade Root Region

At a rotational speed of 9 rpm, the flow at the surface of the blade root exhibits a very complex form. Flow stream separated by generated from the blade root so that pressure difference in the radial direction and centrifugal acceleration of blade move in tip. At a rotational speed of 12.17 rpm, flow separation due to reduced fluid flow contributes to an enhanced output. The blade surface pressure is high owing to the surrounding wind effects, and the pressure is higher along the tips. The blade rotates owing to the lift force generated from the pressure difference between the pressure and suction sides.

The pressure surface of the blade exhibits a high-pressure state as it directly receives the wind. Tips experience increasingly higher pressure. Relative velocity is the resultant of the rotation speed and speed of flow into the blade; therefore, it is unaffected by the angle of attack. At this time, the pressure difference between the pressure and suction sides generates lift and rotates the blade. In order to confirm, this position divided by length of the blade gives the pressure distribution on the airfoil. It can be confirmed that the pressure increases from the root to tip of the blade. At a rotational speed of 3 rpm in the analysis, a negative value of the pressure gradient was observed that leads to the formation of vortex flows. This is shown in Fig. 10.

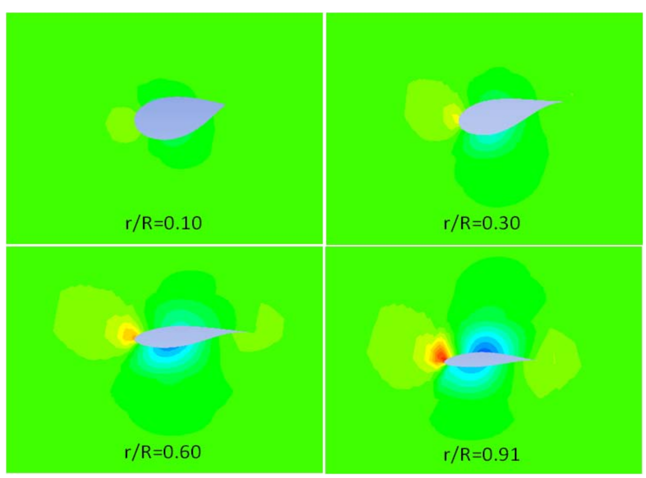

Fig. 9. Pressure Distribution at four cross-sections

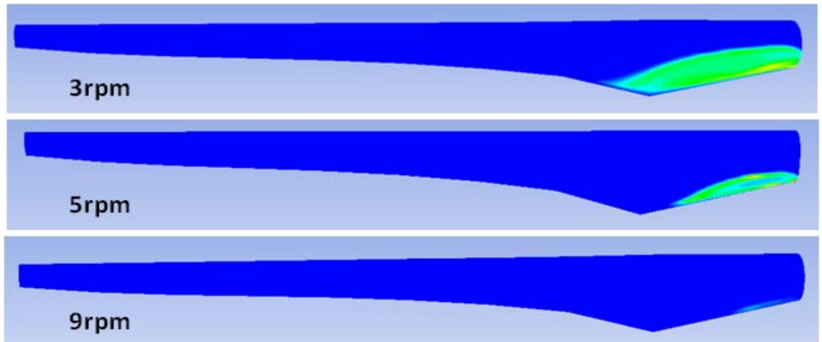

Fig. 10. Turbulence Viscosity Distribution with Change in Rotational Speed

The relation between output graph by rotational speed is shown in Fig. 11. At a rated rotational speed of 12.17 rpm, the output reached approximately 2.6 MW. Finally, the blade power reached the rated output value, so it can be verified that the design optimization was done well within the aerodynamic flow fields.

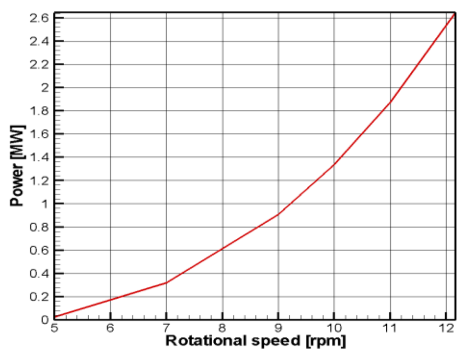

Fig. 11. Power v/s Rotational Speed

Acknowledgment

This work was supported by the Human Resources Development program of the Korea Institute of Energy Technology Evaluation and Planning (KETEP) via a grant funded by the Korean government’s Ministry of Trade, Industry and Energy (No.20113020020010).

References

W.P. Jones and B.E. Launder, (1972), “Prediction of Laminarization with a Two-Equation Model of Turbulence,” International Journal of Heat and Mass Transfer, Vol. 15, No. 2, pp. 301–314.

P.R. Spalart, S.R. Allmaras, (1992), “A One-Equation Turbulence Model for Aerodynamic Flows”, AIAA Paper, No. 92-0439.

J.L. Tangler, D.M. Somers, (1987), "Status of the special purpose airfoil families", Proceedings Conference Windpower.

Y. Chen, Y. Zhiquan, L. Deyuan, and H. Fupeng, (2006), “Numerical simulation of large angle of attack separated flow over airfoils of HAWT rotors”, Wind Engineering, Vol. 30, No. 1, pp. 35-42.

E. Ferrer, X. Munduate, (2007), “Wind Turbine Blade Tip Comparison using CFD”, Journal of Physics: Conference Series, Vol. 75.
H. Aa Madsen, R. Mikkelsen, S. Oye, C. Bak, J. Johansen, (2007), "A Detailed investigation of the Blade Element Momnetum model based on analytical and numerical results and proposal for modifications of the BEM model", Journal of Physics, Vol. 75.
R. Lanzafame, M. Messina, (2007), “Fluid Dynamic Wind Turbine Design: Critical Analysis, Optimization and Application of BEM Theory”, Renewable Energy, Vol.32, pp. 2291-2305.
C.A. Baxevanou, P.K. Chaviaropoulos, S.G. Voutsinas and N.S. Vlachos, (2008), “Evaluations Study of a Navier-Stokes CFD Aeroelastic Model of Wind Turbine Airfoils in Classical Flutter”, Journal of Wind Engineering and Industrial Aerodynamics, Vol. 96, pp. 1425-1443.
D.A. Griffin, T.J. McCoy, (2008), “COE Reductions through Active Aerodynamic Control of Rotor Aerodynamics and Geometry”, Subcontract Report, NREL/SR-500-43391.
D.A. Griffin, T.J. McCoy, (2008), “COE Reductions through Active Aerodynamic Control of Rotor Aerodynamics and Geometry”, NREL Technical Report SR-500-43391.
D. Hartwanger, A. Horvat, (2008), “3D Modeling of a Wind Turbine using CFD”, NAFEMS Conference.
J. Jonkman, S. Buterfield, W. Musial and G. Scott, (2009), “Definition of a 5MW Reference Wind Turbine for Offshore System Development”, NREL Technical Report TP-500-38060.
F. Villalpando, M. Reggio, A. Ilinca, (2011), “Assessment of Turbulence Models for Flow Simulation around a Wind Turbine Airfoil”, Modeling and Simulation in Engineering, Vol. 2011.
S. Rajakumar, D. Ravindran, (2012), “Iterative approach for optimizing coefficient of power, coefficient of lift and drag of wind turbine rotor”, Renewable Energy, Vol. 38, pp. 83-93.