International Journal of Control and Automation
Vol. 9, No. 12 (2016), pp.275-286
http//dx.doi.org/10.14257/ijca.2016.9.12.23

Numerical Simulation on Wind Turbine Airfoil Aerodynamics Performance

Li Jun and Song Wenlong *

College of Mechanical and Electrical Engineering, Northeast Forestry University, Songwl139@163.com

Abstract

The aerodynamic performance of wind turbine airfoils was an important foundation for the aerodynamic design and the performance analysis of the wind turbine. In this paper, fluent aerodynamics performance of a wind turbine airfoil S414 is simulated based on CFD. Its pressure coefficient with different angles of attack was obtained. The numerical results are compared with the experimental data, which proves the simulation data of Spalart-Allmaras equation is fit for airfoil S414. The theoretical foundation is established for three-dimensional flow simulation of airfoil in the future. Besides, adopting Spalart-Allmaras turbulence models can improve the predicting results for aerodynamic performance. The proposed method provides an effective way to predict the aerodynamic characteristics of wind turbine. This study provides quick and valid numerical simulation methods for wind turbine airfoil aerodynamic designing and possesses higher utility value in engineering.

Keywords*: wind turbine; airfoil; numerical calculation*

1. Introduction

Renewable energy studies have been growing rapidly due to rising energy demand, finite fossil fuels and environmental concerns [11]. In today's energy structure, wind energy is a most promising clean energy in renewable energy, wind energy available around the world is much greater than the current world energy consumption. As the rapid development of human society and economy, environmental pollution and energy supply had become more and more prominent, so the development and utilization for the renewable energy, especially for the wind energy has become the world's important issue [12]. Over the last several decades, there have been great efforts paid to investigate how surface roughness affects the performance of wind turbine. Surface roughness in the wind turbine community is a broad term containing different sizes and shapes of roughness, including the almost non-detectable roughness such as dust or slight erosions up to large-scale roughness like some severe erosion defects or ice accretion geometries [23]. The leaf blade, as one of the core components of wind turbine, good aerodynamic performance of the airfoil was the key factors which affected the power coefficient of the wind turbine, so an important part of leaf blade’s aerodynamic design was rational choice for the airfoils. Compared with the developed countries like United States, Sweden, Denmark in the field of wind energy technology, in China, the geometry and aerodynamic performance parameters for dedicated wind turbine airfoils lacked currently [12]. Based on the location of the wind turbines, the blades suffer from various environmental factors that create different blade surface roughness or even defects, forming various airfoil profiles. In recent years, the effects of leading edge erosion or defect on the performance of wind turbine, has jumped into the sight of researchers and engineers and act as a significant negative factor for large scale wind turbines in some challenging environments [24-25], such as offshore regions

ISSN: 2005-4297 IJCA

with severe salt corrosion, desert areas with abrasive sand particles, etc. Though the roles leading edge defect playing in the degradation of wind turbine performance is well known and some initial studies has been conducted, few work has been done to quantify the effects of erosion or defect on wind turbine performance. Previous studies mainly focused on the slight erosion of turbine blade with pits or gouges on the surface [26], or studies with simplified erosion patterns with small amount of variations [24]. Additionally, a majority of these investigations have been conducted through experimental measurements, while the local flow field near the erosion or defect can hardly be presented to demonstrate the influence of leading edge erosion or defect. Therefore, there are still significant gaps in the knowledge of the aerodynamic and flow characteristics for eroded or damaged wind turbine blades. The choice of airfoils mainly included NACA series airfoils, NREL S series airfoils, SERI series airfoils, RISF-A series airfoils, FFAW series airfoils [13] and DU series airfoils. At present, the numerical simulation analysis for the performance of wind turbine airfoils mainly concentrated in the influence of mesh density [14], turbulence model [15-16], leading edge roughness [17], airfoil camber [18] and Reynolds number [19] for the aerodynamic performance. Given the reality that the aerodynamic performance analysis of domestic wind turbine machine mainly based on the numerical simulation, the deep study on the simulation for common airfoils would provide reliable reference for the aerodynamic design of wind turbine machine [12]. Wind turbine is large rotating machinery of external flow, and the flow around the wind turbine blade is quite complex. In the operating environment, due to wind shear, tower shadow effect, gusts, pitch control and yaw regulation, the aerodynamic characteristics of the wind turbine are highly unsteady. So far, the understanding about the unsteady flow mechanism of the wind turbine is very limited [9]. Hence, it is essential to design high performance wind turbine blades so as to increase energy capture and to reduce structural loads [10]. Wind turbine is the center of wind engineering, the performance of wind turbine blades determines the efficiency of using wind energy and airfoil performance determines the efficiency of the blades. At present, the numerical simulation analysis for the performance of wind turbine airfoils [1] mainly concentrated in the influence of mesh density [2], turbulence model [3], leading edge roughness [4], airfoil camber[5] and Reynolds number[6] for the aerodynamic performance. It is the rapid development of computer technology and modern CFD (Computational Fluid Dynamics) technology that brings a great convenience to this simulation work [7].

With the help of the CFD software FLUENT, the method of steady numerical analysis was used in this paper, S414 airfoil which has been designed and analyzed theoretically and verified experimentally in The Pennsylvania State University Low-Speed, Low-Turbulence Wind Tunnel was analyzed and its pressure coefficient with different angles of attack was obtained. The numerical results are compared with the experimental data, which proves the feasibility about our approach to predict aerodynamics performance. The theoretical foundation is established for three-dimensional flow simulation of airfoil in the future.

Unexpected performance degradation occurs in wind turbine blades due to leading edge defect when suffering from continuous impacts with rain drops, hails, insects, or solid particles during its operation life. To assess this issue, this paper numerically investigates the steady and dynamic stall characteristics of an S414 airfoil with various leading edge defects. More leading edge defect sizes and much closer to practical parameters are investigated in the paper.

2. Numerical Methods

All printed material, including text, illustrations, and charts, must be kept within the parameters of the 8 15/16-inch (53.75 picas) column length and 5 15/16-inch (36 picas) column width. Please do not write or print outside of the column parameters. Margins are 3.3cm on the left side, 3.65cm on the right, 2.03cm on the top, and 3.05cm on the bottom. Paper orientation in all pages should be in portrait style.

The numerical method is based on Navier-Stokes equations and introduced by Reynolds [8]. Take incompressible flow as an example, the Reynolds averaged Navier-Stokes equations are expressed as,

Mass equation:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

momentum equation:

$$\frac{\partial}{\partial x_j}(u_j u_i) = -\frac{1}{\rho}\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j}(\mu \frac{\partial u_i}{\partial x_j} - \overline{u_i' u_j'}) \tag{2}$$

Where u is fluctuation average speed; u' is fluctuation instantaneous value; ρ is air density; p is air pressure; x_i is a coordinate; μ is air dynamic viscosity coefficient.

$$\frac{\partial}{\partial x_i}(\tilde{v}\rho u_i) = G_v + \frac{1}{\sigma_{\tilde{v}}}[\frac{\partial}{\partial x_j}\{\mu + \rho\tilde{v})\frac{\partial \tilde{v}}{\partial x_j}\} + C_{b2}\rho(\frac{\partial \tilde{v}}{\partial x_j})^2] - Y_v + S_{\tilde{v}} \tag{3}$$

The control equations were Spalart-Allmaras equations, which only solves the transport equation of kinematic eddy viscosity and doesn't solve the length associated with the shear layer thickness. Therefore, computational resource requirements are low, it is an aerospace application design without flow and it shows obvious good results in the boundary layer with adverse pressure gradient effect. Except in the near wall region, the transport variables are consistent of the turbulent viscosity, the transport equation is following as:

$$\frac{\partial}{\partial x_i}(\tilde{v}\rho u_i) = G_v + \frac{1}{\sigma_{\tilde{v}}}[\frac{\partial}{\partial x_j}\{\mu + \rho\tilde{v})\frac{\partial \tilde{v}}{\partial x_j}\} + C_{b2}\rho(\frac{\partial \tilde{v}}{\partial x_j})^2] - Y_v + S_{\tilde{v}} \tag{4}$$

Where $\tilde{v}$ is turbulent transport variable; G_v is the term of turbulent viscosity; Y_v is the damaged item of turbulent viscosity; $\sigma_{\tilde{v}}$ and C_{b2} are absolute terms; v is kinetic viscosity and $S_{\tilde{v}}$ is self-defining term.

Numerical computation is conducted using the Spalart-Allmaras turbulence model, and the method has been validated by comparison with the SST k-ω turbulence model. In order to ensure the calculation convergence, the residuals for the continuity equation are set to be less than 10^{-7} and 10^{-6} in steady state and dynamic stall cases. The simulations are conducted with the software ANSYS Fluent 13.0. The transport equation of k and ω is following as:

$$\frac{D(\rho k)}{Dt} = P_k - \beta^* \rho\omega k + \partial x_j[(\mu + \sigma^* \mu_T)]\frac{\partial x}{\partial x_j} \tag{5}$$

$$\frac{D(\rho\omega)}{Dt} = \frac{\gamma\omega P_k}{k} - \beta\rho\omega^2 + \frac{\partial}{\partial x_j}[(\mu + \sigma\mu_T)]\frac{\partial x}{\partial x_j}] \tag{6}$$

$$\mu_T = \gamma^* \frac{\rho k}{\omega} \tag{7}$$

The SST k-ω model equation is following as:

$$\frac{D(\rho k)}{Dt} = P_k - \beta^* \rho \omega k + \partial x_j [(\mu + \sigma_k \mu_T) \frac{\partial k}{\partial x_j}] \tag{8}$$

$$\frac{D(\rho\omega)}{Dt} = \frac{\gamma\omega P_k}{\mu_T} - \beta\rho\omega^2 + \frac{\partial}{\partial x_j}[(\mu + \sigma\mu_T)]\frac{\partial\omega}{\partial x_j}] + 2(1 - F_1)\frac{\rho\sigma_{\omega 2}}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial\omega}{\partial x_j} \tag{9}$$

$$\mu_T = \frac{\rho k \alpha_1}{\max(\alpha_1 \omega, \Omega F_2)} \tag{10}$$

Where P_k is turbulent kinetic energy generation item; β^* is a constant 0.09. Other parameter values are showed in reference [20-22].

3. Numerical Calculation

3.1. Creation of Mesh Model

A wind turbine airfoil S414 is chosen in this research. The geometric parameters are same as reference [5], and we use Gambit on S414 airfoil model. Because of air flow differences in the airfoil front and rear, airfoil surface mesh was divided into the front half and the back half. In order to capture the boundary layer flow to meet the requirements of low Reynolds number model.

We employ encryption to the mesh near the wall. The total number of model mesh was 90000, which 400 points distributed on the airfoil surface totally. The first layer mesh height of the wall is 1.0×10-5m and the first grid spacing on the airfoil was determined to make y+ less than 1. In this study, the C-grid topology was adopted. We choose 10 times the airfoil chord as the calculation domain boundary. The grid is gradually stretched from the boundary layer to the calculation domain boundary. Figure.1,2 shows the final meshes.

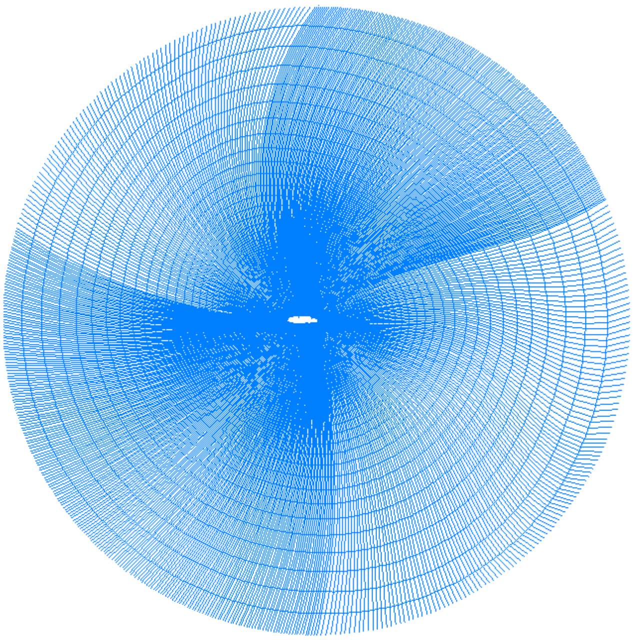

Figure 1. Close View of Mesh Distribution

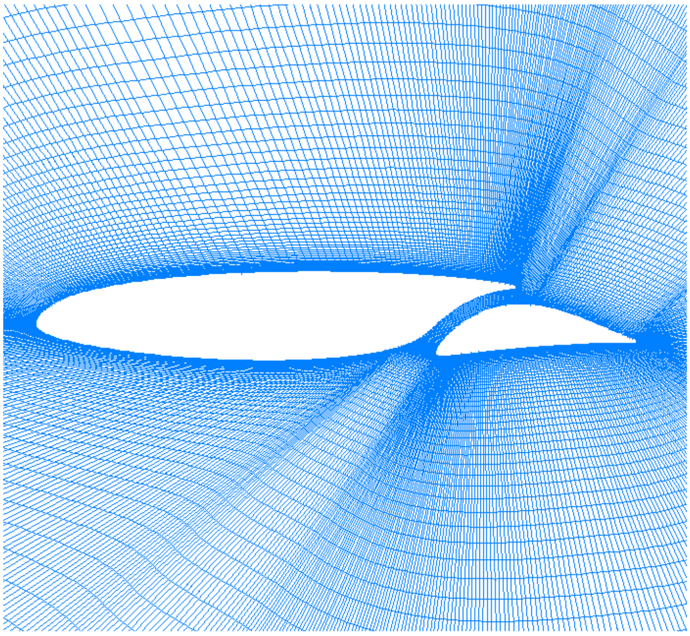

Figure 2. Partial Enlarged View of Mesh Distribution

3.2. Boundary Conditions

The airfoil surface is a no slip surface. Inlet velocity was obtained according to different values of Reynolds number. The calculation of different inlet angles of attack is carried out. That is the same in reference [8].

So, boundary conditions were set as follows:

(1) Airfoil surface was set as adiabatic no-slip wall;

(2) The Reynolds number of calculation is Re=1.0×106

(3) The angle of attack covers a range of 1°, 6.09°, 15.24°.

3.3. Simulation Setup

(1) The steady-state solver based on pressure is used.

(2) The SIMPLEC scheme is used to solve the pressure-velocity coupling.

(3) Momentum and turbulent equation are dispersed by QUICK format.

(4) Calculation error is 10-5~10-4.

4. Results and Analysis

4.1. Contour of Streamline and Pressure Coefficient

Figure 3 Figure 4 and Figure 5 show contour of streamline and pressure coefficient on the projected attack angle -1.04°, 6.09°, 15.24° at Airfoil S414. Airfoil surface flow is the attached flow in small angles, with angle increasing, the flow occurs the separation trend.

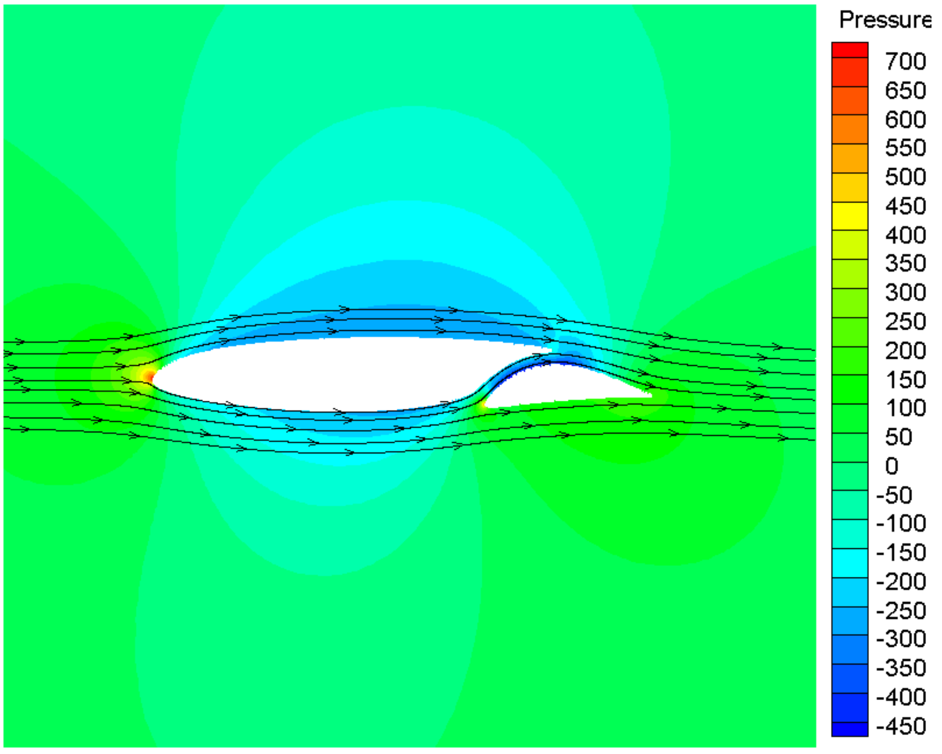

Figure 3. Contour of Streamline and Pressure Coefficient of Attack Angle 1.04°

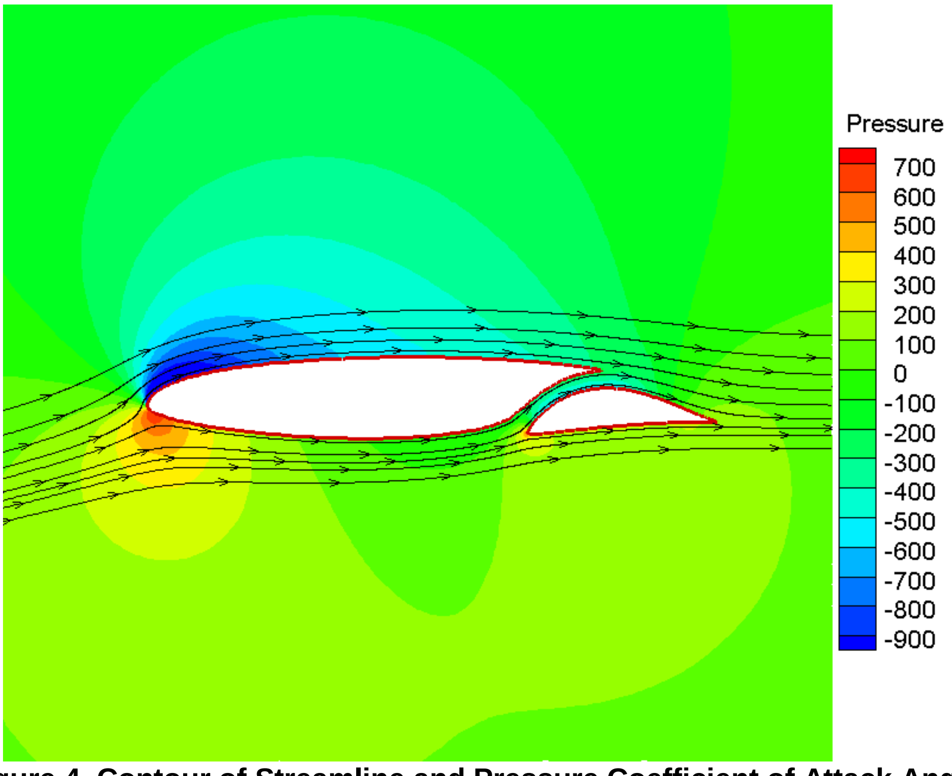

Figure 4. Contour of Streamline and Pressure Coefficient of Attack Angle 6.09°

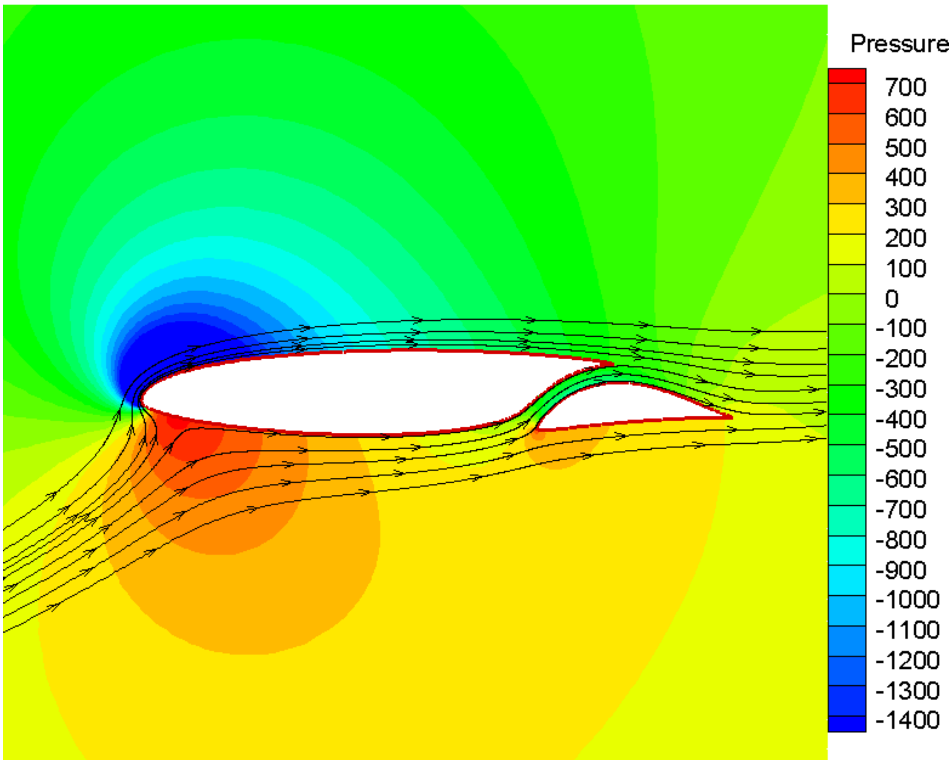

Figure 5. Contour of Streamline a Pressure Coefficient of Attack Angle 15.24°

4.2. Comparison of Simulation and Experiment Results

The comparison of the theoretical and experimental pressure distributions for the airfoil shape at various angles of attack for a Reynolds number of 1.00×10^{6} and a Mach number of 0.10 with transition free is shown in Figure. 6, 7, 8. C in X-axis represents the airfoil chord, Y-axis C_p represents pressure coefficient and C_p can be approximated by:

$$C_p = \frac{p - p_\infty}{\frac{1}{2}\rho u_\infty^2} \tag{11}$$

As shown in Figure. 6, 7, 8. that both airfoil S414 and its fitted airfoil exhibit smooth continuity in free transition and fixed transition cases, except for the back blade about the attack angle of -1.04°. The pressure gradient along the majority of the upper surface of the fore element becomes flat (Figure. 6). As the angle of attack is increased even further, the pressure peak on the upper surface of the fore element becomes sharper (Figure. 7). The maximum lift coefficient occurs at an angle of attack of 16.24° (Figure. 8). At small angles (Figure. 6, 7), which is near the middle of the low-drag range, the agreement between the predicted and measured pressure coefficients and pressure gradients is good. The predicted location of the laminar separation bubble on the upper surface of the aft element is slightly aft of the measured location. At an angle of 16.24° (Figure. 8), the agreement is less precise, particularly with respect to the pressure gradients on the upper surface of the fore element. The predicted locations of the laminar separation bubbles on the upper surfaces of the fore and aft elements are aft of the measured locations.

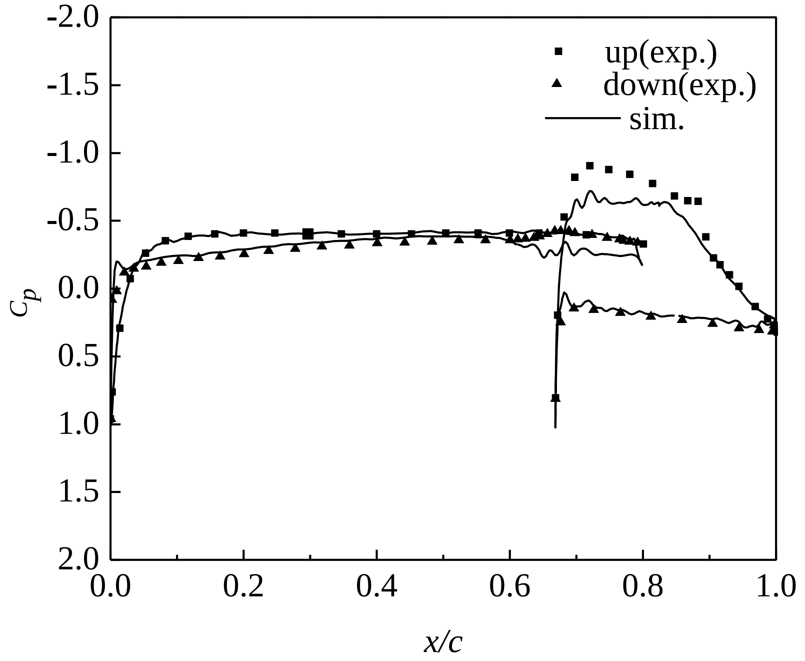

Figure 6. Pressure Distributions of Attack Angle -1.04°

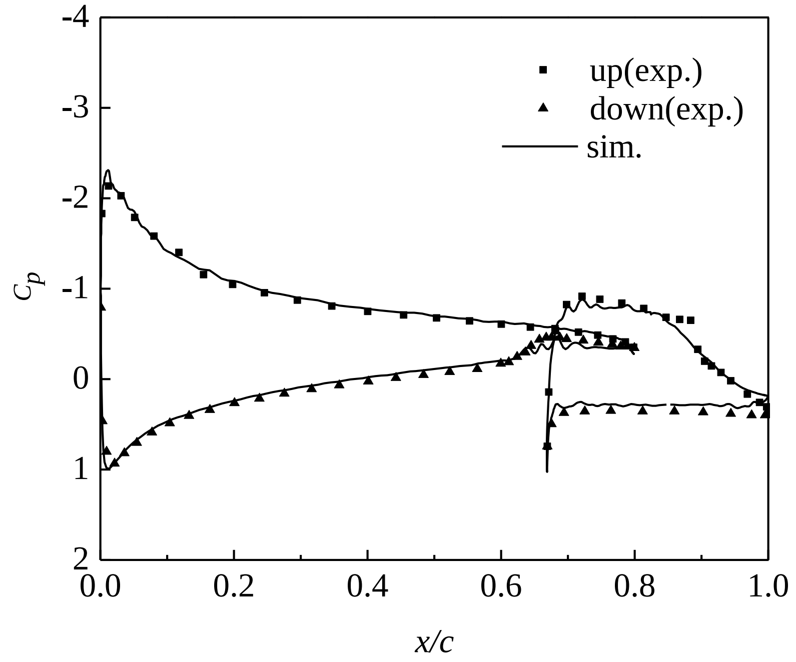

Figure 7. Pressure Distributions of Attack Angle 6.09°

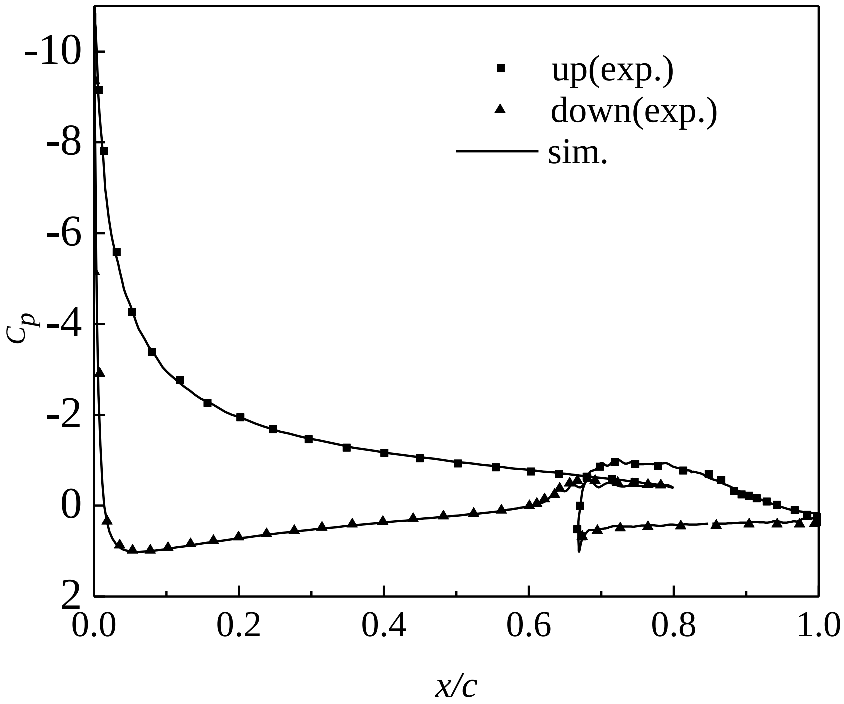

Figure 8. Pressure Distributions of Attack Angle 15.24°

Figure 8 and figure 9 show the comparison of attack angle -1.04°, 6.09° pressure coefficient based on different speeds 24m/s、36m/s、48m/s with Spalart-Allmaras and SST k-ω model.

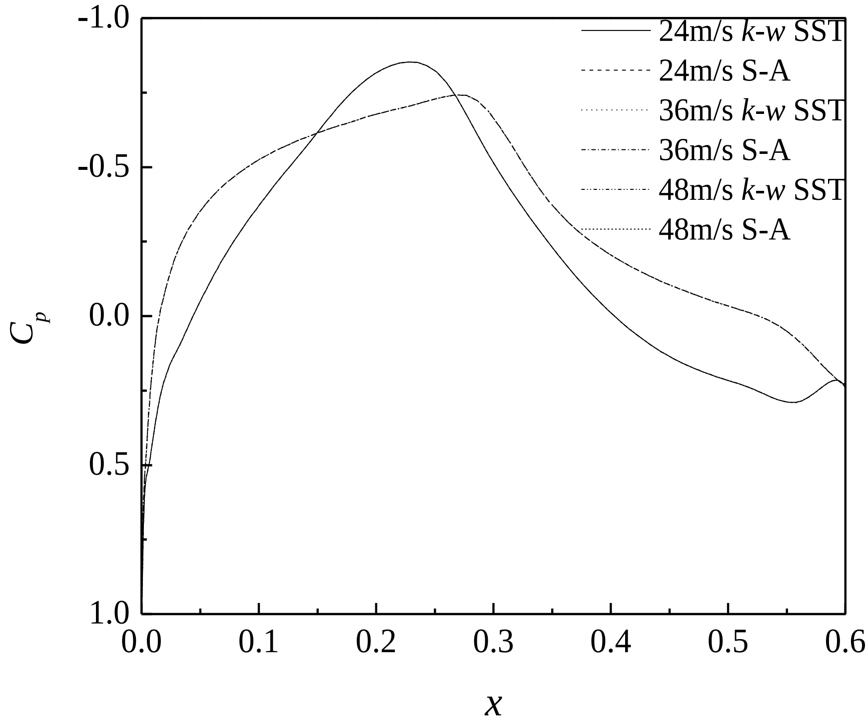

Figure 9. Comparison of Pressure Coefficient Based On Different Speed (-1.04°)

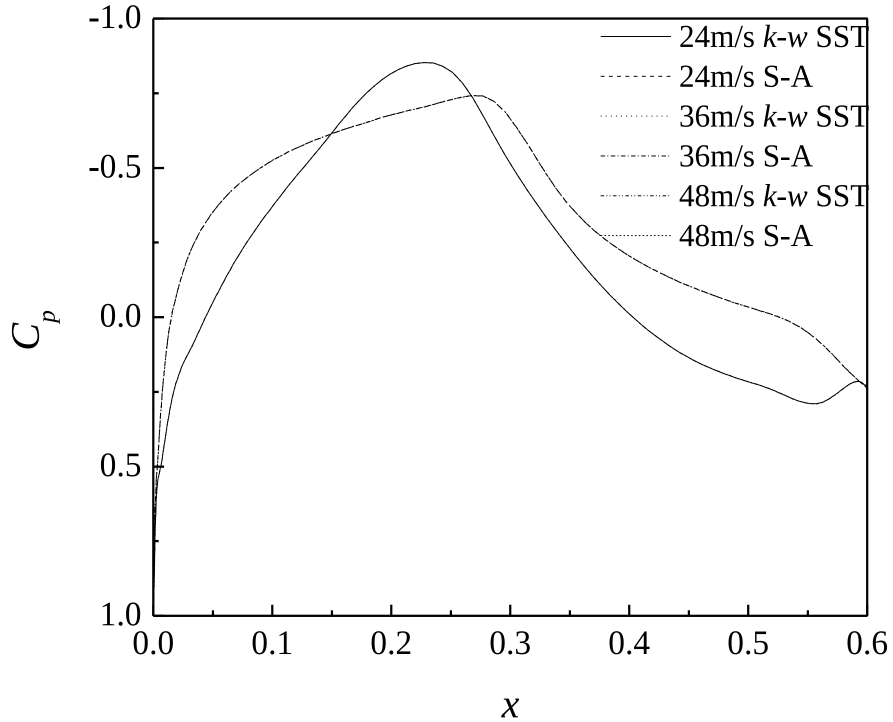

Figure 10. Comparison of Pressure Coefficient Based On Different Speed (6.09°)

Obviously, the simulation results of -1.04°and 6.09°are basically coincident with different speeds, so we can conclude that the inlet velocity has little influence on the pressure coefficient of the upper and lower surfaces. In this paper, we choose the Spalart-Allmaras model to simulate the main operating conditions.

5. Conclusions

This study presents dynamic overset CFD simulations for wind turbine airfoil S414. Apart from the fact that this test campaign generated an experimental data bank that is currently being implemented in a wind turbine simulation code, it also made it possible to estimate the aerodynamic properties of a wind turbine airfoil subjected to high levels of turbulence. The results indicate that our method is feasible for predication of the wind turbines airfoil aerodynamics performance. In the future, we can extend the two dimensional numerical simulation to three-dimensional flow.

Acknowledgments

This research was sponsored by Natural Science Foundation of Heilongjiang Province of China (No. QC2013C035), the Fundamental Research Funds for the Central Universities of China (No. 2572014CB13) and Foundation of Heilongjiang Educational Committee (12533014).

References

[1] M. Ismail and K. Vijayaraghavan, "The effects of aerofoil profile modification on a vertical axis wind turbine performance", Energy, vol. 80, **(2015)**, pp. 20-31.

[2] L. Ma, J. Chen, G. Du and R. Cao, "Numerical simulation of aerodynamic performance for wind turbine", ACTA ENERCIAE SOLARlS SINICA, vol.2, **(2010)**, pp: 203-209.

[3] L. Liu and J. Z. Xu, "The effects of turbulence model on the aerodynamic prediction of wind turbine blade", Journal of engineering the ENGINEERING THERMOPHYSICS, vol.7, **(2009)**, pp. 1136-1139.

[4] P. Chen, J. P. Liu and W. M. Zhang, "Performance prediction and analysis of wind turbine airfoil", ACTA ENERCIAE SOLARlS SINICA, vol. 10, **(2009)**, pp: 1424-1428.

[5] R. N. Li, S. A. Zhang, R. Yang and D. S. Li, "Effect of Aerofoil Camber on Airfoil Aerodynamic Performance", FLUID MACHINERY, vol. 37, no. 5, **(2009)**, pp:17-21.

[6] G. Y. Zhang, W. M. Feng, C. L. Liu and J. F. Yu, "Simulation study on Blade Design and Aerodynamic Function of Wind Turbine", ENERGY RESEARCH & UTILIZATION, vol.1, **(2005)**, pp. 21-25(in Chinese).

[7] A. Ebrahimi, M. Hajipour and H. Hasheminasab, "Experimental Investigation on the Aerodynamic Performance of NLF-0414 Iced-Airfoil", JOURNAL OF APPLIED FLUID MECHANICS, vol. 9, no. 2, **(2016)**, pp: 587-592

[8] O. Reynolds, "On the dynamical theory of turbulent incompressible viscous fluids and the determination of the criterion", Phil. Trans. R. Soc. London A, vol.186, **(1894)**, pp. 123-161.

[9] L. Liu, "Research on the Unsteady Aerodynamic Characteristics of Wind Turbine Blades", Institute of Engineering Thermophysics Chinese Academy of Sciences, **(2012)**.

[10] J. Chen, Q. Wang, X. P. Pang, S. L. Li and X. F. Guo, "Improvement of airfoil design using smooth curvature technique", Renewable Energy, vol. 51, **(2013)**, pp. 426-435.

[11] R. Rafiee, M. Tahani and M. Moradi, "Simulation of aeroelastic behavior in a composite wind turbine blade", Journal of Wind Engineering and Industrial Aerodynamics, vol. 151, **(2016)**, pp: 60-69.

[12] J. Yaoa, W. B. Yuan, J. L. Wang, J. B. Xie, H. P. Zhou, M.J. Peng and Y. Sun, "Numerical simulation of aerodynamic performance for two dimensional wind turbine airfoils", International Conference on Advances in Computational Modeling and Simulation, vol. 31, **(2012)**, pp: 80-86.

[13] L. Nikolaou, S. Politis and K. Chaviaropoulos, "Modeling the flow around airfoils equipped with vortex generators using a modified 2D Navier – Stokes solver", Journal of Solar Energy Engineering, vol. 127, **(2005)**, pp: 223-233.

[14] J. D. Anderson, "Dynamics and application of computational fluid", Mechanical Industry Press, **(2007)**.

[15] L. J. Ma, J. Chen, G. Du and R. J. Cao, "Numerical simulation of aerodynamic performance for wind turbine", ACTA ENERCIAE Solarls SINICA, vol. 2, **(2010)**, pp: 203-209.

[16] L. Liu and J. Z. Xu, "The Effects of Turbulence Model On the Aerodynamic Performance Prediction of Wind Turbine Blade", Journal of Engineering Thermophysics, vol. 7, **(2009)**, pp: 1136-1139.

[17] P. Chen, J. P. Liu and W. M. Zhang, "Performance Prediction And Analysis Of Wind Turbine Airfoil", Acta Enerciae Solarls Sinica, vol. 10, **(2009)**, pp. 1424-1428.

[18] R. N. Li, S. A. Zhang, R. Yang and D. S. Li, "Effect of Aerofoil Camber on Airfoil Aerodynamic Performance", FLUID MACHINERY, vol. 37, no. 5, **(2009)**, pp. 17-21(in Chinese).

[19] G. Y. Zhang, W. M. Feng, C. L. Liu and J. F. Yu, "Simulation study on Blade Design and Aerodynamic Function of Wind Turbine", Energy Research & Utilization, vol. 1, **(2009)**, pp. 21-25(in Chinese).

[20] F. Menter, "Two-equation Eddy Viscosity Turbulence Models for Engineering Applications", AIAA Journal, vol. 32, **(1994)**, pp: 1299-1310.

[21] D. C. Wilcox, "Turbulence modeling for CFD", Canada: DCW Industries, **(1993)**.

[22] P. Fuglsang and C. Bak, "Development of the Riso Wind Turbine Airfoils", Wind Energy, vol. 7, no. 2, **(2004)**, pp: 145-162.

[23] W. W. Huebsch, A. P. Rothmayer, "Numerical prediction of unsteady vortex shedding for large leading-edge roughness", Computers & fluids, vol. 3, no. 33, **(2004)**, pp: 405-34.

[24] K. Gharali and K, D. A. Johnson, "Numerical modeling of an S809 airfoil under dynamic stall, erosion and high reduced frequencies", Applied Energy, vol. 93, **(2012)**, pp: 45-52.

[25] H. Slot, E. Gelinck, C. Rentrop and E. Heide, "Leading edge erosion of coated wind turbine blades: Review of coating life models", Renewable Energy, vol. 80, **(2015)**, pp: 837-48.

[26] A. Sareen, C. A. Sapre and M. S. Selig, "Effects of leading edge erosion on wind turbine blade performance", Wind Energy, vol. 10, no. 17, **(2014)**, pp. 1531-42.

Authors

Li Jun, she received the BS degree from Liaoning Technical University(2000), and received the MS degree from Harbin Engineering University(2006), and received PHD degree from Harbin Institute of Technology(2010). She is currently an associate professor in College of Mechanical and Electrical Engineering, Northeast Forestry University. Her research interests are in the areas of computer application.

Wenlong Song, he received the BS degree, the MS, the PHD degree from Northeast Forestry University (1995, 2004, 2008). He is currently a professor in College of Mechanical and Electrical Engineering, Northeast Forestry University. His research interests are in the areas of pattern recognition and intelligent system and information detection.