

Aerodynamic Characteristics Analysis of Blade Airfoil

Zhang Li-da, Mao Jin-duo, Chen Rong-sheng, Lian Ling-jun, Wang Xu

XHU Wind Power Technology Institute, Chengdu 610039, China

Abstract: In this article, we analyzed the airfoil FFA-W3-211 by fluid dynamics software, and we get the figure of lift, drag coefficient, figure of lift-drag ratio, and relationship of pressure in the aerofoil surface with the variation of attack angle. Finally, we analyze these figures, and get the aerodynamic performance of FFA-W3-211.

Key words: Airfoil, Aerodynamic characteristic, Mesh generation, Turbulence model

CLC number: TK8

Document code: A

1. Introduction

Wind is a kind of energy, which is clear, renewable and has rich reserves. As gradually shortage of energy, many countries have put great financial and labor supports into the exploitation and utilization of wind-power. Nowadays, wind power generation is the main way of wind power utilization. The key word to use wind energy in a high efficiency way is to reduce cost and increase benefit of wind energy generation^{[1][2]}. Wind turbine use blade to converse wind energy, and the characteristic of the blade airfoil directly affect the efficiency of wind turbine. To study the character of blade, we have to study the character of airfoil first^[3].

2. Control Equation

Fluid flow satisfies the law of conservation of mass, the law of conservation of energy and the law of conservation of momentum, and in hydrodynamics these laws embody themselves in the Continuity Equation, the N-S Equation and the Energy Equation. These equations are fundamental equations in Numerical Calculation, and the solution module of N-S Equation system is core part of the soft CFD^[4]. 2-dimension concrete expression of these equations as follows:

2-dimension Continuity Equation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_x)}{\partial x} + \frac{\partial(\rho u_y)}{\partial y} = 0 \dots\dots\dots (1)$$

2-dimension N-S Equation

$$\begin{aligned} \frac{\partial(\rho u_x)}{\partial t} + u_x \frac{\partial(\rho u_x)}{\partial x} + u_y \frac{\partial(\rho u_x)}{\partial y} &= -\frac{\partial \rho}{\partial x} + \mu \left(\frac{\partial^2 u_x}{\partial x^2} + \frac{\partial^2 u_x}{\partial y^2} \right) \\ \frac{\partial(\rho u_y)}{\partial t} + u_x \frac{\partial(\rho u_y)}{\partial x} + u_y \frac{\partial(\rho u_y)}{\partial y} &= -\frac{\partial \rho}{\partial y} + \mu \left(\frac{\partial^2 u_y}{\partial x^2} + \frac{\partial^2 u_y}{\partial y^2} \right) \end{aligned} \quad (2)$$

2-dimension Energy Equation

$$\begin{aligned} \frac{\partial(\rho e)}{\partial t} + u_x \frac{\partial(\rho e)}{\partial x} + u_y \frac{\partial(\rho e)}{\partial y} &= \\ -\rho \left(\frac{\partial u_x}{\partial x} + \frac{\partial u_y}{\partial y} \right) - \frac{2}{3} \mu \left(\frac{\partial u_x}{\partial x} + \frac{\partial u_y}{\partial y} \right)^2 &+ \mu \left[2 \left(\frac{\partial u_x}{\partial x} \right) + \left(\frac{\partial u_y}{\partial x} + \frac{\partial u_x}{\partial y} \right) + 2 \left(\frac{\partial u_y}{\partial y} \right)^2 \right] \end{aligned} \quad (3)$$

3. Turbulence Model

In fluid viscous movement, the most complex is turbulence flow, it presented as endless scale in physical feature and strong nonlinear in mathematical description, and there has no complete theory to explain flow mechanism by now. Though turbulence flow can be described by N-S Equation, no computer is able to solve complex turbulence problem in engineering practice by solving N-S Equations at present.

Turbulent filed can be described in a relatively better way by solving average N-S Equations, but it needs to introduce turbulence model to close the equations. $k-\varepsilon$ tow-equation turbulence model is widely used but isotropic is its presumption and hypothesis. $k-\varepsilon$ tow-equation turbulence model has a simple form and it can give higher calculation accuracy, but it cannot reflect the influence of isotropic and average vortex. And then a new model theory- RNG $k-\varepsilon$ model based on renormalization group method has been took out. In the form RNG $k-\varepsilon$ model is the same with standard $k-\varepsilon$ model, the difference is the coefficient. Turbulence constant in this model is obtained by renormalization group theory, and in this model introduce a coefficient to modified turbulent energy dissipation rate. Compare with standard $k-\varepsilon$ model, RNG $k-\varepsilon$ model has a higher accuracy in problems that has high average strain rate^[5-7]. The expressions as follow:

$$\begin{aligned} \frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) &= \frac{\partial}{\partial x_j} \left(\alpha_k \mu_{eff} \frac{\partial k}{\partial x_j} \right) + G_k + G_b - \rho \varepsilon - Y_M + S_k \\ \frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) &= \frac{\partial}{\partial x_j} \left(\alpha_\varepsilon \mu_{eff} \frac{\partial \varepsilon}{\partial x_j} \right) + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} - R_\varepsilon + S_\varepsilon \end{aligned} \quad \dots (4)$$

Therefore, in this paper N-S Equations is closed with RNG $k-\varepsilon$ model.

4. Mesh Generation

Mesh Generation of numerical simulation of the flow field is the basis for solving equations, and it will have a direct impact on the numerical results. The division of flow field can be structured or unstructured, both have advantages and disadvantages. Structured grid data

is simple, and the grid generation is faster with good quality. But its typical shortcoming is only can be used in a narrow scope. Non-grid structure makes up for the disadvantage of the grid, that is can not solve any arbitrary shape and connectivity of the regional grid. But the grid generation is heavy workload, and discrete equations solving speed is slower.

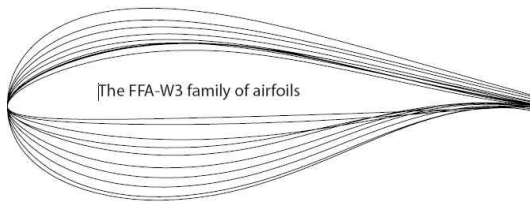


Fig.1 the FFA-W3 family of airfoil



Fig.2 airfoil FFA-W3-211

The FFA-W3-211 airfoil, its geometry is not complicated. Taking the efficiency of numerical solution into consideration, structured grid is used in the calculation. In addition, C-type mesh is often used in division of the Grid with airfoil mesh. Numerical condition used in numerical simulation is free flow $Ma = 1.2$, $Re = 1 \times 10^6$. The situation of final grid is airfoil surface grid layout 200 points; flow grid computing is divided into three domains, respectively as a 200×100 region and two 100×100 symmetrical regions. The computational domain calculates a total of 40,400 nodes and 40,000 grids.

Numerical regional border includes import border, export border and from top to bottom airfoil surface border. Boundary condition sets for the far-field pressure on the border and the roughness of the non-slip surface 0 boundary condition.

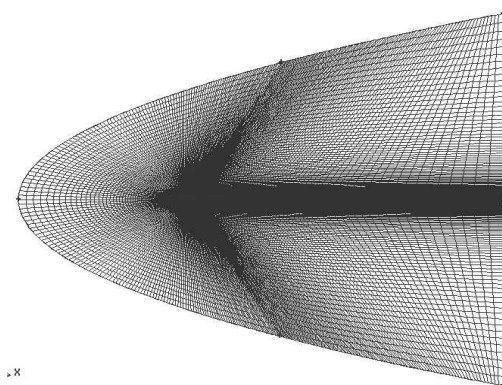


Fig.3 mesh of flow field

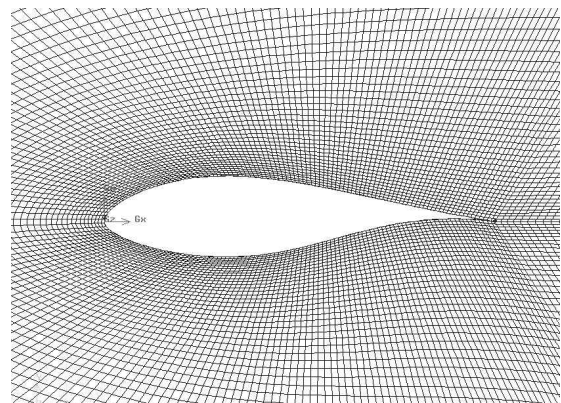


Fig.4 mash around the airfoil

5. Analysis of Results

In graph-5, we can get that, when $\alpha < 6^\circ$, both lift force and drag force increase with the attack angle α . When $\alpha = 16^\circ$, the lift force get to the maximum, and the lift coefficient is about 1.5; after that, as the attack angle α grow, the lift force become to decrease, but the drag fore increases quickly, and aerodynamic performance of the airfoil become badly; when α grows to 20° , lift force decrease to minimum value, and lift coefficient is about 1.0; when $\alpha > 20^\circ$, with growing of α , the airfoil get into depth stall area, and lift force has a lightly increase. The analysis give out that when $Re = 1 \times 10^6$, the stall angle of airfoil FFA-W3-211 is about 16° .

Lift-drag ratio reflects the aerodynamic quality, the bigger the ratio is, the better the airfoil

performed. In graph-6, maximum value of the ratio is 60 and the attack angle is $\alpha=7^\circ$. When $7^\circ < \alpha < 20^\circ$, lift-drag ratio decreases quickly with α , and when $\alpha > 20^\circ$, downward trend of the ratio become slow, and when $\alpha=30^\circ$, lift-drag ratio is only about 4.

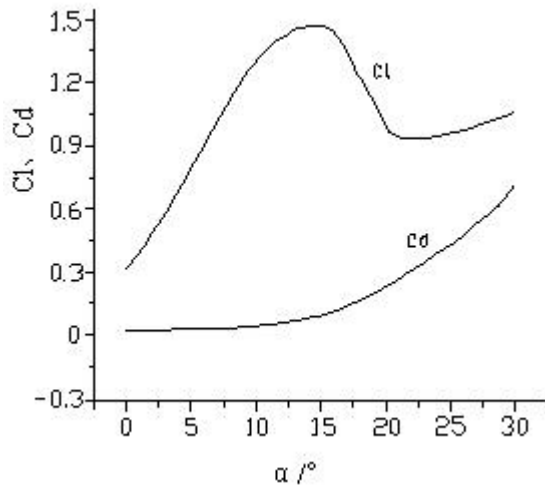


Fig.5 curves of lift and drag coefficient

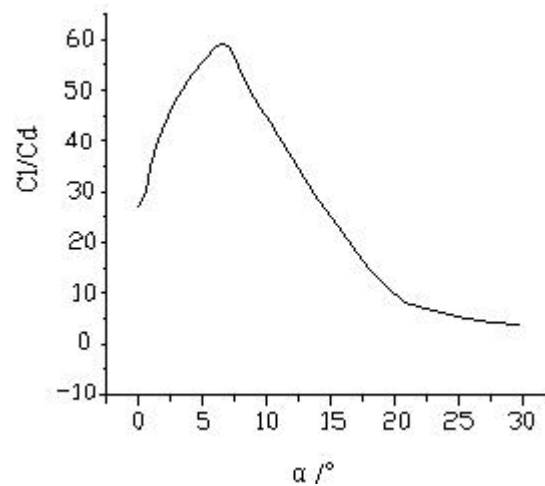
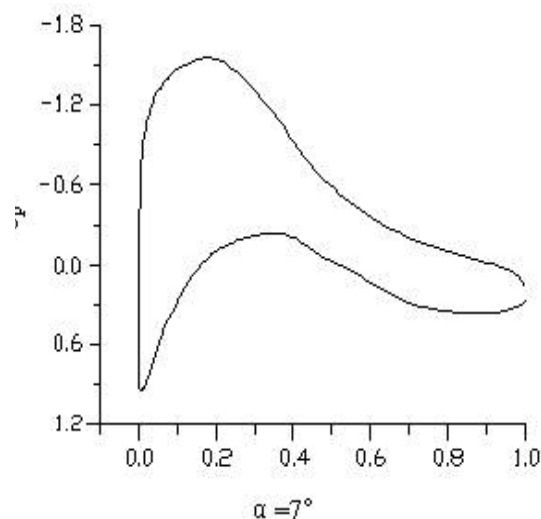
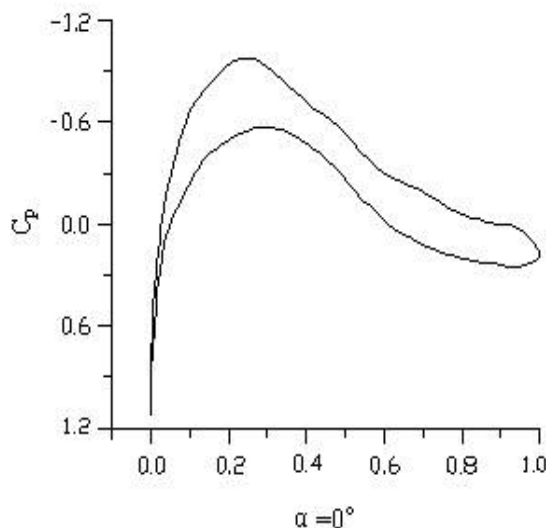


Fig.6 curves of lift-drag ratio

Graph-7 is the pressure coefficient of the airfoil, it reflect the flow characteristics at different attack angle. When the flow is occurred at $\alpha < 7^\circ$, high pressure range is only at the front of the airfoil, the lift force is at a low level. When $7^\circ \leq \alpha \leq 16^\circ$, pressure at the pressure part is at a high level and absolute value of the pressure at the suction surface is still at a big value, so the pressure difference is high and it can provide a high lift force. But, at the same time, the high absolute pressure part at the front of the airfoil suction surface become to decrease. When $16^\circ < \alpha \leq 20^\circ$, absolute pressure part at the suction surface decrease quickly, but increasing of the pressure at the pressure part is slow, the pressure difference decreased with the attack angle, so the lift force is decreased too. When $\alpha > 20^\circ$, absolute pressure part at the suction surface keeps invariant, reasure at the pressure part has a slow increase, so the lift force get a bit upturn.



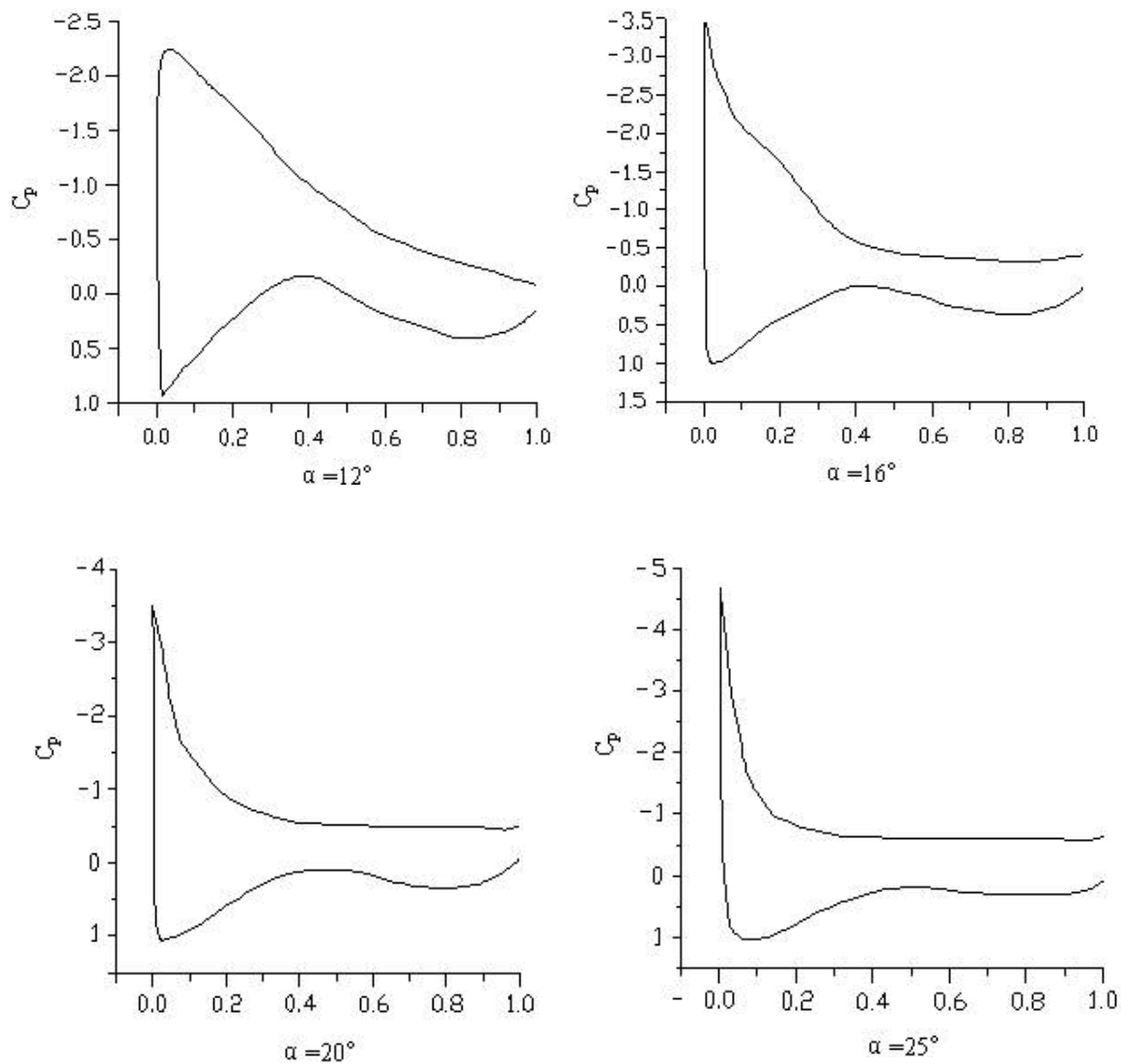


Fig.7 pressure coefficient around the airfoil surface

6. Ending Words

In this article, we analyzed the airfoil FFA-W3-211 by fluid dynamics software, and we get the figure of lift, drag coefficient, figure of lift-drag ratio, and relationship of pressure in the aerofoil surface with the variation of attack angle. The whole flow field Mesh Generation use C structured grid and in the calculation $RNGk - \varepsilon$ model is used, and that give a result, simulation of the flow filed is similar to real. In this paper, Surface Roughness of the airfoil is not considered, Surface Roughness should be considered in the future work.