Available online at www.sciencedirect.com
SciVerse ScienceDirect
Procedia Engineering 31 (2012) 213 – 219
Procedia Engineering
www.elsevier.com/locate/procedia

International Conference on Advances in Computational Modeling and Simulation

Analysis on the aerodynamic performance of vertical axis wind turbine subjected to the change of wind velocity

Huimin Wang[a*], Jianliang Wang[a], Ji Yao[a], Weibin Yuan[b], Liang Cao[a]

[a]*Faculty of Civil Engineering and Architecture, Kunming University of Science and Technology, Kunming 650024*
[b]*Faculty of Civil Engineering and Architecture, Zhejiang Institute of Technology, Hangzhou 310014*

Abstract

Reynolds averaged Navier-Stokes equations and Realizable $k-\varepsilon$ model were used in this paper, and the two dimensional unsteady flow field of the vertical axis wind turbine was simulated numerically at different wind velocity. The calculation results showed that the velocity in the region of wind turbine's rotation was much larger than the air flow of the upstream. The length of the wind turbine's downstream wake dispersion region was increased with the increase of the wind velocity. There is a much larger value of the eddy in the rear region of the wind turbine's rotational blades. And eddy existed in the downstream region of the wind turbine, and the larger velocity of cross flow, the larger value of the downstream flow's eddy. When the rotational speed was constant, with the increase in wind velocity, the variation of the wind turbine's total torque coefficient tended to smooth. The calculation results pointed out the direction for the follow-up study.

Keywords: vertical axis wind turbine; Realizble $k-\varepsilon$ turbulence model; wind velocity; torque coefficient

1. Preface

With the continued rapid development of economy, energy problem in China has become more and more critical. The contradictions between power structure composed mainly by thermal power and the increasing depletion of fossil energy resources become increasingly acute, the development and utilization of alternative energy have become issues need to be resolved for China's strategy of sustainable development. Wind power as a renewable clean energy, receives highest attention from the governments, the energy sector and the environmental communities around the world. So far, the analysis

*Corresponding author. Tel.: +86-13700691860;
E-mail address: whuimin@vip.sina.com

doi:10.1016/j.proeng.2012.01.1014

methods for wind turbine's aerodynamic performance include blade element momentum theory, eddy wake method and computational fluid dynamic method. With the recent rapid development of computer hardware and software technology, CFD moves away from the constraints of the computer calculation capacity, and being applied widely in engineering because of its advantage as low cost, high computing speed, complete information and no limit for the scale of models. FLUENT is one of the most comprehensive, the most widely applicable, the most widely domestic used CFD software. In the U.S. market, the share of FLUENT is 60%. Basically, as long as related to fluid problems in engineering, all can be solved with FLUENT. Therefore, a two-dimensional model of a vertical axis wind turbine[1] is established in this paper, with the help of FLUENT, the unsteady numerical calculation[2-4] for the vertical axis wind turbine is carried out with sliding grid technology. And the flow field characteristics, the torque variation and wind energy utilization of the vertical axis wind turbine in a complete rotation cycle are analyzed under the change of the wind speed.

2. Calculation theory and numerical simulation method

2.1. Control equation

As the wind turbine's Mach number at work is typically less than 0.3, so the flow around the airfoil could be considered as incompressible flow, the two dimensional incompressible N-S equations and two dimensional continuity equation are used as the control equations.

The two dimensional incompressible N-S equation is expressed in vector as follows:

$$\rho \frac{DV}{Dt} = \rho \boldsymbol{f} - \nabla p + \mu \nabla^2 \boldsymbol{V} \tag{1}$$

Where $\boldsymbol{V}$ is the velocity vector, $\boldsymbol{f}$ is the volume force vector, μ is the dynamic viscosity.

The continuity equation is showed as follows:

$$\nabla \cdot V = 0 \tag{2}$$

2.2. Turbulence model

Compared with the conventional $k-\varepsilon$ turbulence model, the content on rotation and curvature is added to the Realizable $k-\varepsilon$ model for the calculation of turbulence dynamic viscosity, and the equation of the dissipation rate ε is amended. Therefore, the model been shown that could simulate the flow around a blunt body effectively. Reynolds averaged equations and continuity equations are showed as follows:

$$\frac{\partial \overline{u_i}}{\partial t} + \overline{u_j} \frac{\partial \overline{u_i}}{\partial x_j} = -\frac{\partial \overline{P}}{\partial x_i} + \frac{\partial}{\partial x_j}[\nu \frac{\partial \overline{u_i}}{\partial x_j} - \rho \overline{u_i u_j}] \tag{3}$$

$$\frac{\partial \overline{u_i}}{\partial x_i} = 0 \tag{4}$$

The mode equations of the turbulent kinetic energy k and turbulent kinetic energy dissipation rate ε are:

$$\frac{\partial k}{\partial t} + \overline{u_j} \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_i}[(\nu + \frac{\nu_t}{\sigma_t}) \frac{\partial k}{\partial x_j}] + (\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i}) \frac{\partial \overline{u_i}}{\partial x_j} - \varepsilon \tag{5}$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial(\rho\overline{u_i}\varepsilon)}{\partial x_i}=\frac{\partial}{\partial x_i}[(\nu+\frac{\nu_t}{\sigma_\varepsilon})\frac{\partial\varepsilon}{\partial x_i}]+c_1\rho S\varepsilon-c_2\rho\frac{\varepsilon^2}{k+\sqrt{\nu\varepsilon}} \quad (6)$$

Where $\overline{u_i}(i=1,2)$ are average velocity component in the x,y direction separately; $\overline{P}$ is pressure; ν is the dynamic viscosity coefficient of the fluid; ρ is the density of the air; $\nu_t=C_\mu k^2/\varepsilon$ is the viscosity coefficient of the eddy group.

Compared with the standard $k-\varepsilon$ model, the biggest modification of Realizable $k-\varepsilon$ model is the amendment of the coefficient C_μ, which takes the time-average strain rate into account. The other coefficients in the model are taken as follows:

$$\sigma_\varepsilon=1.2, \quad \sigma_t=1.0, \quad c_2=1.92, c_1=\max(0.43,\frac{\eta}{1+\eta}), \eta=\frac{Sk}{\varepsilon}, S=(2S_{i,j}S_{i,j})^{1/2} \quad (7)$$

$$S_{i,j}=\frac{1}{2}(\frac{\partial\overline{u_i}}{\partial x_j}+\frac{\partial\overline{u_j}}{\partial x_i}) \quad (8)$$

Realizable $k-\varepsilon$ model has been widely used in various types of flow simulation, which includes rotational average shear flow, free flow including jet and mixed flow, flow in pipes, boundary layer flow and backward-facing step flow, the result is consistent with the experimental data.

2.3. Numerical method

A certain type of vertical axis wind turbine use three blades of H type, airfoil of NACA0018, the length of chord is C=0.1m, the diameter of wind turbine is 0.9m, and rotational speed is 100r/min. the calculation domain of the model is showed in Figure 1, the range for the domain is x_{min}=-1.6m， x_{max}=+4m， y_{min}=-1.6m， y_{max} =+1.6m.

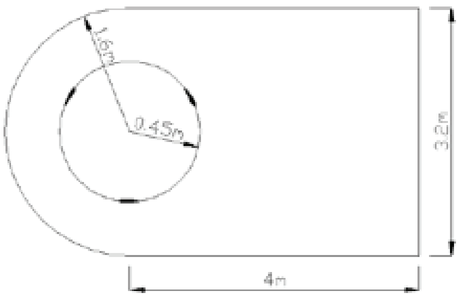


Figure 1 diagram of the model

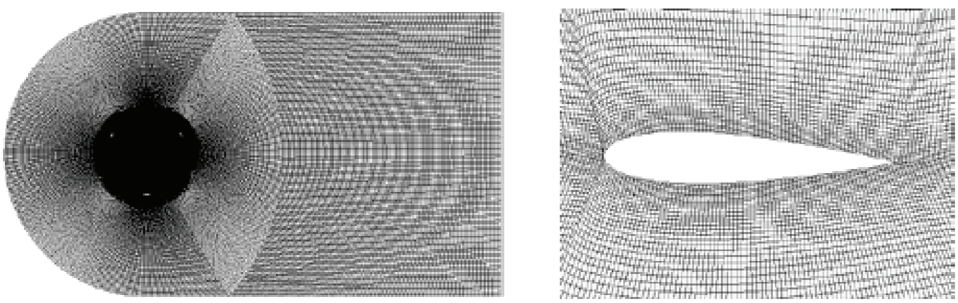


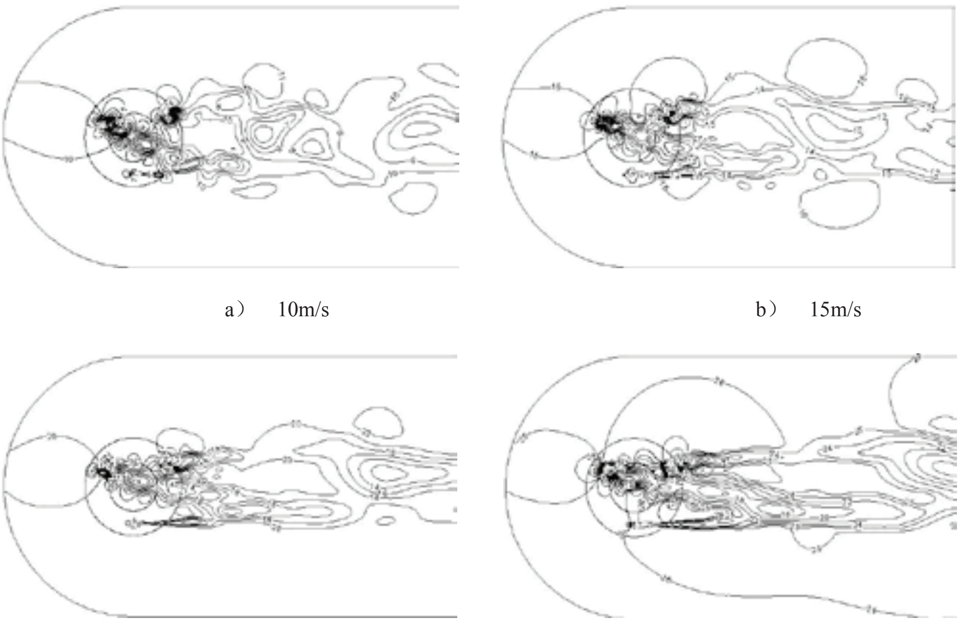
Fig.2.calculation domain and grids around the blade:a) grids of the calculation domain; b) grids around the blade

The separation solver of FLUENT is used, the component of pressure, velocity, turbulent kinetic energy, dissipation rate and Reynolds stress adopt second-order upwind difference scheme. And SIMPLEC algorithm is applied for the coupling of pressure and velocity, with the help of ICEM software, a geometric model is established. Finite volume element method is applied, the calculation region is divided into 116598 grids, the calculation region and the grids around the blade are showed in Figure 2. The inlet boundary on the left adopts the conditions of velocity inlet, according to the different wind velocity, there are 4 calculation working conditions. The first case has the wind velocity of 10m/s, the second case is 15m/s, the case is 20m/s and the fourth case is 25m/s. The turbulence intensity of inlet flow is set to 6% at one atmosphere pressure, the outflow boundary on the right adopt fully developed turbulent conditions. The wall is the blade's surface, no-slip adiabatic boundary conditions are used, and standard wall function is applied when close to the wall[5]. Considering that the calculation domain is divided into rotation part and static part, so sliding grid technology is used, that means the interface boundary is defined as sliding surface. The time step is 0.001, and Realizable $k-\varepsilon$ model is used. In the solution process, the residuals for variables are monitored, as well the moment of force coefficient, to determine whether the convergence of the solution. The convergence of the residuals is set as 10^{-4}.

3. Analysis of the numerical simulation results

After the calculation process is stable, the calculation results of four cases are given in this paper.

3.1. Contour of velocity



a） 10m/s b） 15m/s

c） 20m/s d） 25m/s

Figure 3 velocity contour at different wind velocity

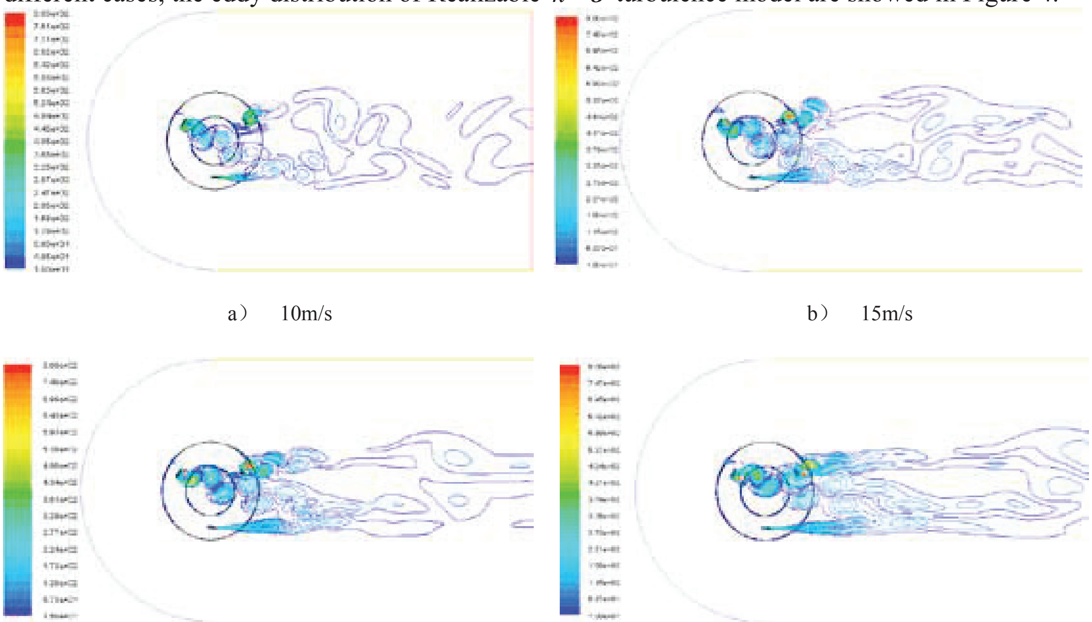
After the calculation process is stable, at the same time and the same rotational speed (n=100r/min), in different working conditions, the velocity contour of Realizable $k-\varepsilon$ turbulence model are showed in Figure 3.

It can be seen from Figure 3 that, the velocity in the region of wind turbine's rotation was much larger than the air flow of the upstream by means of the flow field distribution at the same wind velocity. However, with the wake away from the wind turbine, the velocity of wake approaches the velocity of upstream air flow. This phenomenon is mainly caused by the existence of gradient between the wake velocity and the velocity of downstream free air flow. Then shear turbulence will occur between the two flows, and momentum will exchange between them, finally the velocity difference between the wake and free air flow will disappear.

The flow fields at different wind velocity are compared, then it can be seen from Figure 3 that the length of the wind turbine's downstream wake dispersion region was increased with the increase of the wind velocity, at the same time, the trend for the increase of wake's length rises too. This reflects the influence of the wind velocity on the downstream flow field.

3.2. Distribution of eddy

After the calculation process is stable, at the same time and the same rotational speed (n=100r/min), in different cases, the eddy distribution of Realizable $k-\varepsilon$ turbulence model are showed in Figure 4.



a） 10m/s

b） 15m/s

c） 20m/s

d） 25m/s

Figure 4 distribution of eddy

It can be seen from Figure 4 that, for four cases, the eddy is much larger around the wind turbine's blade, especially in the upper blade's leeward of the wind turbine's rotational part. This indicates that the flow field around the blade in this region is complex, and eddy of large scale exist around the blade. While eddy is less in the lower blade's back of the wind turbine's rotational part, this indicates that adhesion flow mainly exist around the blade. Eddy exists in the downstream region of the wind turbine's rotational part, which indicate the influence of the wind turbine's blade to the flow field of downstream. For four cases, the larger velocity of inlet wind, the larger eddy of the downstream flow field.

3.3. Change of torque coefficient

The torque coefficient C_M is used as the performance evaluation, in order to show that from the rotational force generated by the wind, the amount of wind turbine gains for the available moment of force.

$$C_M = \frac{M}{0.5\rho A V_\infty^{\ 2} R} \qquad (9)$$

Where M ($N \cdot m$) is the torque of the wind turbine; V_∞ (m/s) is the wind velocity of inlet flow; ρ is the density of air, taken as 1.225kg/m3 at standard atmosphere pressure; A is the area, and R (m) is the radius of the wind turbine.

After the calculation process is stable, the curve of the wind turbine's total torque changed with different time for four cases are showed in Figure 5. It can be seen from Figure 5 that, in a cycle of 0.6s, the total torque coefficient of the vertical axis wind turbine for four working conditions all has a clear periodicity. This is the significant difference between the vertical axis wind turbine with the horizontal axis wind turbine. At the same rotational speed, the first case (wind velocity of 10m/s) has the biggest negative total torque coefficient, while the second case (wind velocity of 10m/s) has the biggest positive total torque coefficient. At the same rotational speed, with the increase of the wind velocity, the variation of the wind turbine's total torque coefficient tends to smooth.

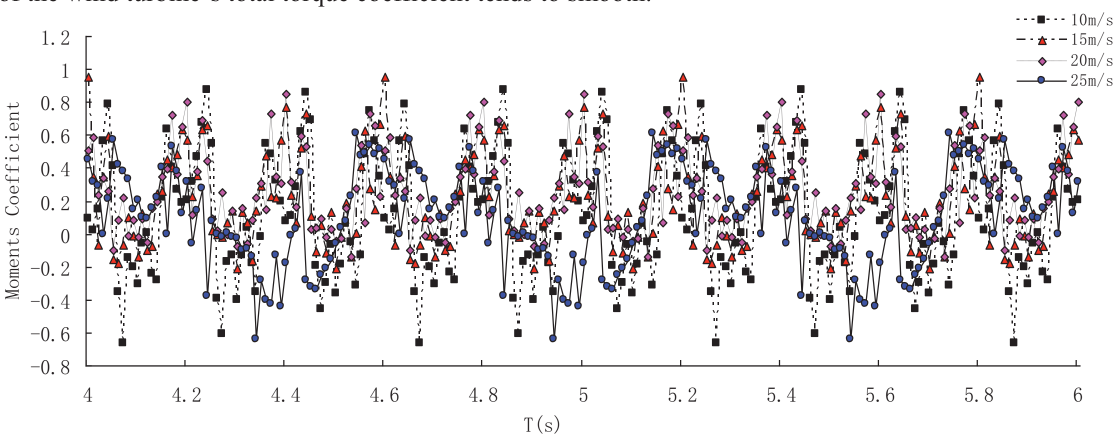


Figure 5 total torque coefficient diagram changed with time

4 conclusion

The two dimensional model of the vertical axis wind turbine is established in this paper, and the two dimensional unsteady flow field was simulated numerically at different wind velocity. The analysis results show that: (1) the velocity in the region of wind turbine's rotation was much larger than the air flow of the upstream. There is a wake dispersion region in the downstream of the wind turbine, and the length of the wake dispersion region was increased with the increase of the wind velocity. (2) the eddy is much larger in the upper blade's back of the wind turbine's rotational part, while eddy is less in the lower blade's back of the wind turbine's rotational part. Eddy also exists in the downstream region of the wind turbine's rotational part, and the larger velocity of inlet wind, the larger eddy of the downstream flow field. (3) At the same rotational speed, the condition of lower wind velocity has larger total torque coefficient. With the increase of the wind velocity, the variation of the wind turbine's total torque coefficient tends to smooth.

Acknowledgements

This work was financially supported by major social development project in science and technology projects of Science and Technology Department of Zhejiang Province (2009C13012).

Reference:

[1] Akiyoshi Iida Akisato Mizuno,Keiko Fukudome Numerical Simulation of Aerodynamic Noise Radiated form Vertical Axis wind Turbines[C]. Proceedings of 18 international Congress on Acoustics,2004, CD-ROM.

[2] Zhang Guoyu, Feng Weimin, Liu Changlu, Yu Jianfeng. Simulation study on Blade Design and Aerodynamic Function of Wind Turbine [J]. ENERGY RESEARCH & UTILIZATION. 2009(1):21-25(in Chinese).

[3] LG NIKOLAOU，E S POLITIS，P K CHAVIAROPOULOS. Modeling the flow around airfoils equipped with vortex generators using a modified 2D Navier – Stokes solver [J].Journal of Solar Energy Engineering,2005,127:223-233

[4] Zhao wei guo,Li ren nian,Li de shun,Yang cong xin,zhang yu liang. Choice of Turbulence Model for Numerical Simulation of Special Airfoil of Wind Turbine[J]. JOURNAL OF XIHUA UNIVERSITY(NATURAL SCIENCE EDITION) ，2007，26（6）：61-62(in Chinese).

[5] Huang Jian-feng , Zhang Li-xiang, He Shi-hua. Numerical Simulation of 3-D Steady and Unsteady Flows in Whole Flow Passage of a Francis Hydro-turbine[J]. Proceedings of the CSEE，2008，29(2):87-84(in Chinese).