Available online at www.sciencedirect.com

SciVerse ScienceDirect

Procedia Engineering 31 (2012) 245 – 250

Procedia Engineering

www.elsevier.com/locate/procedia

International Conference on Advances in Computational Modeling and Simulation

Analysis on the influence of rotational speed to aerodynamic performance of vertical axis wind turbine

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

[a]*Faculty of Civil Engineering and Architecture, Kunming University of Science and Technology, Kunming, 650024, China*
[b]*College of Architecture and Civil Engineering, Zhejiang University of Technology,Hangzhou,310032,China ;*

Abstract

A two dimensional vertical axis wind turbine's model was established in this paper, and two dimensional unsteady incompressible N-S equations and Realizable $k-\varepsilon$ turbulence model were solved with software FLUENT. SIMPLC algorithm was applied, combined with the sliding grid technology; the influence of rotational speed to the flow structure of vertical axis wind turbine was discussed. The results showed that, the rotation of wind turbine had significant influence on wake, and higher the rotational speed, the greater reduction of the wake velocity. The wake velocity restored gradually away from the rotational part. There was much larger turbulent kinetic energy near the tail of the wind turbine's blade. The value of turbulent kinetic energy reduced gradually away from the rotational part, and the flow restored the stratospheric state gradually. With the increase of wind turbine's rotational speed, the value of turbulent kinetic energy in calculation domain increased too. The results showed that the flow structure of vertical axis wind turbine's rotational process could be revealed effectively by numerical simulation, provided theoretical reference for the engineering design of the vertical axis wind turbine.

Keywords: Rotational speed; Vertical axis wind turbine; Realizable $k-\varepsilon$ turbulence model; Turbulent kinetic energy ;

1. Preface

With the further development of the economy, the demand for energy, particularly for electricity is increasing rapidly. Taking into account the reality that the social power of 80% provided by the

*Corresponding author.Tel.:+86-15198726267.
E-mail address: 27444791@qq.com.

doi:10.1016/j.proeng.2012.01.1019

conventional fossil energy[1], the depleting of current conventional energy sources and protection of the global environment makes human society facing the enormous challenges. So the development and utilization for wind energy which is clean, renewable and abundant received more and more attention around the world. Wind turbine is the key device to wind energy's conversion, there are two main forms for current wind turbines: horizontal axis wind turbine and vertical axis wind turbine according to the location of the wind and the wind turbine's wheel shaft. The vertical axis wind turbine has advantages as simple blade structure and not being affected by the direction of air flow, but so far, many questions still exist in the design for the vertical axis wind turbine's aerodynamic performance. With the development of the computational fluid dynamics and the computer technology, analyzing the aerodynamic performance of vertical axis wind turbine with CFD method is possible. Compared with the theoretical analysis and experimental methods, CFD method [1] has the advantages as short period of study, low cost, all of the flow information can be stored up to study at any time, the observation and display of the information can be controlled initiatively, the simulation of actual scale can be carried out. Through the CFD simulation calculation and analysis, the vertical axis wind turbine's aerodynamic performance [2] [3] can be analyzed and improved efficiently, and can provide necessary basis for the aerodynamic design of the vertical axis wind turbine.

2. Computing Theory and numerical simulation method

2.1. Control equations

As the wind turbine's Mach number at work is generally less than 0.3, so assumed the flow field of the wind turbine as incompressible flow, then the two-dimensional incompressible N-S equation and two-dimensional continuity equation are used as the control equations.

The two-dimensional incompressible N-S equation is shown as follows:

$$\frac{\partial\left(\rho u_i\right)}{\partial t}+\left(\rho u_i u_j\right)_{,j}=-P_{,i}+\mu u_{i,ij} \qquad (1)$$

And two-dimensional continuity equation as follows:

$$\left(\rho u_i\right)_{,i}=0 \qquad (2)$$

Where u_i 、 u_j （i 、 j =1,2） are the velocity components of x, y direction respectively; x_i 、 x_j （i 、 j =1,2） are each coordinate component; P is the average pressure of the flow; μ is the dynamic viscosity coefficient of the fluid and ρ is the fluid density.

2.2. 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, the 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.

Reynolds averaged equations and continuity equations are:

$$\frac{\partial \overline{u_i}}{\partial t}+(\overline{u_i}\,\overline{u_j})_{,j}=-\overline{P}_{,i}+\nu\overline{u}_{i,jj}-\rho(\overline{u_i u_j})_{,j} \qquad (3)$$

$$\overline{u}_{i,i}=0 \qquad (4)$$

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

$$\frac{\partial k}{\partial t}+(k\overline{u_j})_{,j}=((\nu+\frac{\nu_t}{\sigma_t})k_{,i})_{,i}+(\overline{u_{i,j}}+\overline{u_{j,i}})\overline{u_{i,j}}-\varepsilon \qquad (5)$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+(\rho\overline{u_i}\varepsilon)_{,i}=((\nu+\frac{\nu_t}{\sigma_\varepsilon})\varepsilon_{,i})_{,i}+c_1\rho S\varepsilon-c_2\rho\frac{\varepsilon^2}{k+\sqrt{\nu\varepsilon}} \qquad (6)$$

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; t is time; $\nu_t=C_\mu k^2/\varepsilon$ is the viscosity coefficient of the eddy group. c_1 and c_2 are the detail calculations equations of each term in former equations can refer to specific reference.

2.3. Numerical simulation

2.3.1 Object for calculation and grids design

The object for calculation is 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 radius of installation is 0.45m, wind velocity is 15m/s . the calculation domain of the model is showed in Figure 1, x-direction along the horizontal direction, is positive to the right, and y-direction along the vertical direction, is positive upward. The range of calculation is x_{min}=-1.6m，x_{max}=+4m，y_{min}=-1.6m，y_{max} =+1.6m. Blade N.1, BladeN.2 and Blade N.3 are relatively one of the wind turbine's blades, as shown in Figure 2. The grids of wind turbine are showed in Figure 3. Combined with the actual situation of the H type three-blade vertical axis wind turbine, divide the calculation region into fixed part and sliding part, and mesh it in the way of partition mesh, as well as refining grids around the blade in the way of coarse and refine partition. The number of grids in calculation domain is 116598 and the number of elements is 114909 by tests. The calculation is carried out by commercial CFD software FLUENT. The component of pressure, velocity, turbulent kinetic energy and dissipation rate adopt second-order upwind difference scheme. There are three wording conditions according to different rotational speed. The first case is 50r/min, the second is 100r/min and the third is 150r/min.

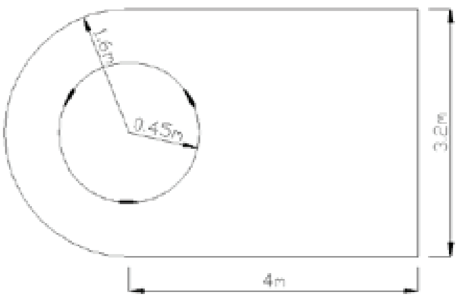


Figure 1 diagram of model

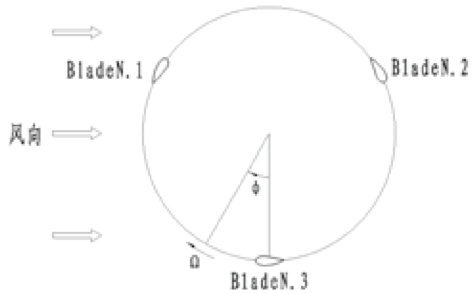
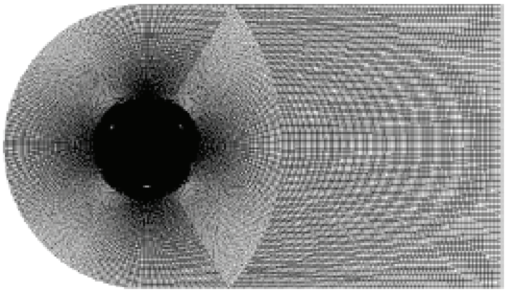
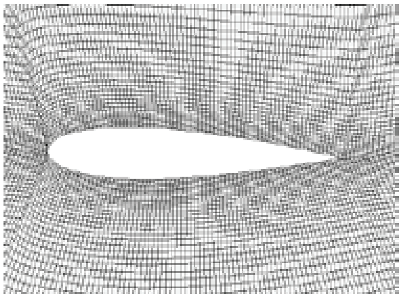


Figure 2 status on wind turbine



a) Grids in calculation domain



b) grids around blade

Figure 3 grids in calculation domain and around blade

2.3.2 Initial conditions and boundary conditions

The initial condition is set as the results come from the model's steady calculation, the time step in unsteady calculation is 0.001s. The external flow field of the wind turbine is showed in Figure 1, and the boundary condition[4] is set as: 1) the velocity inlet condition is applied on the inlet boundary; 2) the fully developed turbulence condition is applied on the outlet boundary; 3) the wall is the blade's surface, defined as no-slip wall, and standard wall function is used to handle the viscous layer of the wall; 4) sliding grid technology is used on the sliding surface, the interface between the grids of the rotational part and the static part is the interface boundary, set as interface.

3. Analysis on the results of numerical simulation

The calculation results of three working conditions after the calculation is stable are given in the paper.

3.1. Velocity distribution of wake

After the calculation is stable, the wake velocity distributions at three sections for different working condition and the same position of wind turbine are intercepted. The section Ⅰ-Ⅰ is 0.60m away from the centre of rotation, section Ⅱ-Ⅱ is 1.5m and section Ⅲ-Ⅲ is 3.0m. The section diagram is showed as Figure 4. The velocity distribution in x direction at three sections in different wording conditions are shown in Figure 5. It can be seen from it that, for three wording conditions, the wind velocity change significantly in the middle of wind turbine, and the wind velocity is close to the inlet velocity in both ends of calculation domain. Compared Figure 5（a）,（b） and（c）, the velocity in x direction restores gradually away from rotation part. This reflects the convection between wake and free flow. It can be seen from Figure 5 (a) that, at the section Ⅰ-Ⅰ, the velocity reduction of the first wording condition is the most small in the middle of wind turbine, mainly caused by the lowest rotational speed of the first wording condition.

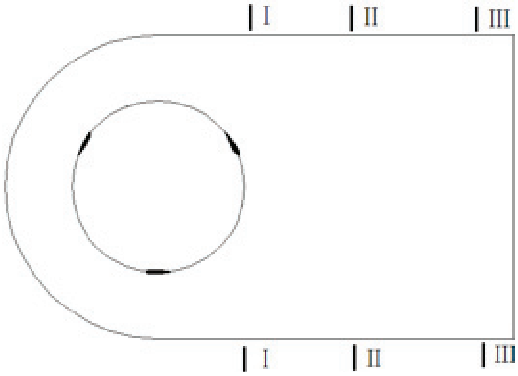
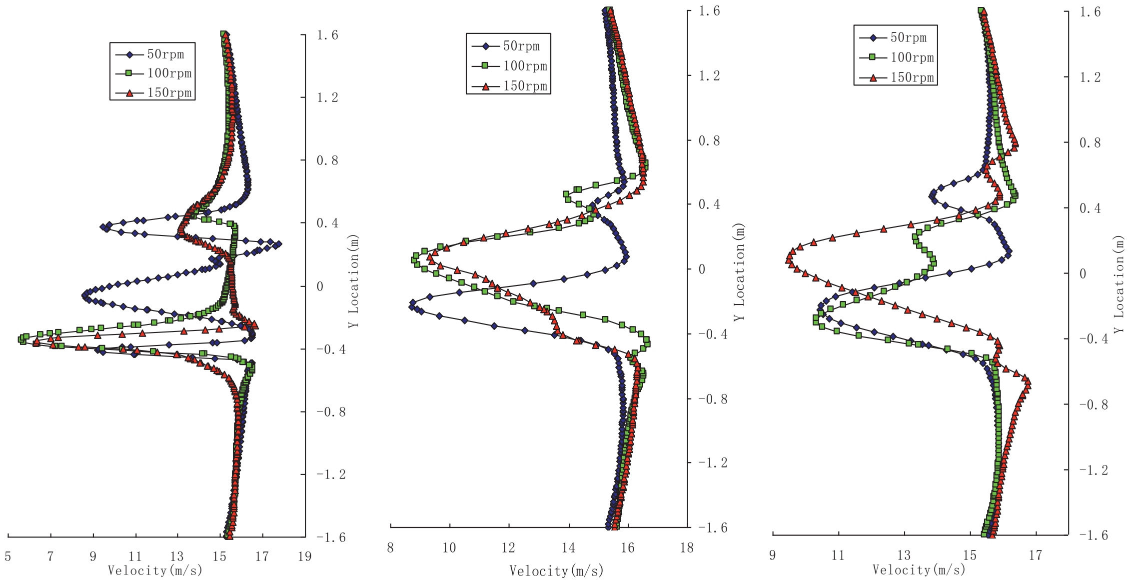


Figure 4 diagram of section

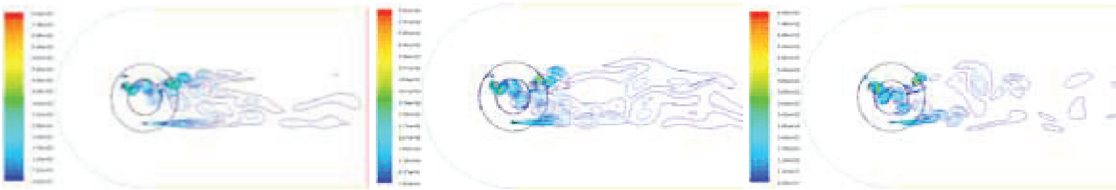


（a）section Ⅰ-Ⅰ （b）section Ⅱ-Ⅱ （c）section Ⅲ-Ⅲ

Figure 5 section’s velocity distribution in x direction

3.2. Distribution of turbulent kinetic energy

After the calculation is stable, the turbulent kinetic energy distributions for different working conditions and the same position of wind turbine are showed in Figure 6. It can be seen from it that the turbulent kinetic energy near the tail of blade in three working conditions are all large, indicates that the flow in this region is in the condition of complex turbulent motion, that is second order flow such as reflux and slip flow exist in the air flow. For all of three working conditions, the upper part of wind turbine’s rotation region has large turbulent kinetic energy and the lower part has little turbulent kinetic energy. There is also the distribution of turbulent kinetic energy in the downstream of wind turbine’s rotation region, which indicates the influence of wind turbine’s rotation on the flow downstream. The value of turbulent kinetic energy reduces gradually away from the rotation region, it indicates that the eddy structure in flow field dissipates gradually, the air flow restores to advection gradually. Compared Figure 6(a) , (b) and (c), the value of turbulent kinetic energy in calculation domain increase with the increase of the wind turbine's rotational speed, which indicates the influence of rotational speed to the flow field structure.



a）case 1 b） case 2 c）case 3

Figure 6 distribution of turbulent kinetic energy （m^2/s^2）

4. Conclusion

The calculation and analysis with the help of CFD, is an important means for understanding the aerodynamic performance of vertical axis wind turbine in design. 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 rotational speed. The analysis results show that: (1) the rotation of wind turbine affects the wake significantly, the higher rotational speed, the larger reduction of the wake's velocity. The wake velocity restores gradually away from the rotation region. (2) The turbulent kinetic energy near the tail of blade are large, indicates that the flow in this region is in the condition of complex turbulent motion. The upper part of wind turbine's rotation region has large turbulent kinetic energy and the lower part has little turbulent kinetic energy. (3) The value of turbulent kinetic energy reduces gradually away from the rotation region, the air flow restores to advection gradually. (4) The value of turbulent kinetic energy in calculation domain increase with the increase of the wind turbine's rotational speed.

Acknowledgement

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

References

[1] John D.Anderson. translated by Songping Wu, Zhaomiao Liu. Dynamics and application of computational fluid [M]. Mechanical Industry Press, 2007,6.

[2] ZHANG Guo-yu, FENG Wei-min,LIU Chang-lu,YU Jian-feng. Simulation study on Blade Design and Aerodynamic Function of Wind Turbine[J]. ENERGY RESEARCH & UTILIZATION,2009, (1) :21-25(in Chinese).

[3] Yang Congxin Wu Faming Zhang Yuliang. Numerical Simulation on Unsteady Rotated Flow of a Vertical Axis Wind Turbine Based on Moving Meshes[J].TRANSACTIONS OF THE CHINESE SOCIETY FOR AGRICULTURAL MACHINERY.2009, 40(6) :98-102.

[4] 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).