Study on Numerical Simulation of Wind Turbine

S Yan, Y T Guo, J Y Zhao, S P Shi and X Liu

Huaneng Clean Energy Research Institute, Beijing102209, China
E-mail: yanshu@hnceri.com

Abstract. Wake effect may decrease the overall power output of the wind farm. It is necessary to study the wind turbine wake, which would benefit wind turbine micro-sitting selection optimization. In order to improve the accuracy of wind turbine wake simulation, dynamic mesh method based on CFD (computational fluid dynamics) is proposed. Simulation comparison tests between dynamic meshing method and actuator disk method have been done with a 1.5MW wind turbine. The result shows that with the effect of the rotating blades being considered, dynamic meshing method can capture the characteristics such as vortexes, velocity distribution, flow recovery rate, and the wake radius along the axis direction of wind turbine wake more accurately.

1. Introduction
The rotor deserves the most concern in a wind turbine, as it can largely influence the power output and efficiency. As wind passes through the rotating blades, its direction and speed can both be changed, and this phenomenon is called the wake effect of wind turbines. In most cases, the wake effect of one wind turbine may cast negative impacts on another one located down the stream, including fatigue load, service life, structural reliability and power output, which will further reduce the total output of the whole wind farm. As a result, more intensive studies of wake effect are in need, with the intention of investigating the wind flow filed, predicting power output, making better use of wind source, reducing the land scale of wind farms, and optimizing the location of wind turbines.

The study of wind turbine wake has a long history. Early works mainly focused on the theoretical analyze of hydromechanics. As the application of CFD became more common, researchers began to use aerodynamic theories and computational simulation in their works. As a result, studies on more complicated cases, with larger and more turbines, were carried out. Risø National Laboratory [1] combined finite volume theory with characteristics of wind farms, in order obtain the aeromechanic performance of a number of turbines when they are arranged tandem or parallel, as well as the velocity filed under different cases. Barthelemie R J [2] introduced a disk, in replacement of the rotor, to simulate the whole wind farm, and got the contour plots of momentum, pressure and velocity. Alexandros Makridis [3] simulated two rows of wind turbines in complex terrains with RSM model and k-wSSSt model, finding out the contour of wind velocity when two turbines are placed over mountain slopes. Zhu Chong [4] did numerical simulation on NH1500 models with general actuator method, discussed the feasibility and got the proper distance of two tandem arranged wind turbines, in order to avoid wake effect. Based on Simplec method and SST k-w turbulence model, Zhou Yunlong [5] simulated the three-dimensional aeromechanic performance of wind turbines and found out that as the number of blades increases, power output and efficiency also see an increase; turbulence becomes more violent, while the total stability gets worse.

As is list above, recent researches only focus on the numerical modeling of individual or multiple wind turbines in different dimensional scales, with static meshes, intrinsically. However, there’s a lack
of enough studies on flow characteristics themselves, especially with dynamic meshes. As a result, this paper, using dynamic mesh method, carries out simulation on 1.5MW wind turbines and compares the result with that of actuator disk model.

2. Numerical Simulation Method

2.1. Dynamic mesh model

2.1.1. Creating single turbine model and dynamic meshes

This paper carries on numerical simulations of a 1.5MW wind turbine on FLUENT15.0, while the blade design and optimization is based on Matlab [6]. The computing model is a cuboid area, with enough inner space for the wake to develop. The length of the area is calculated along Y-axis to represent the flow direction. The width is measured along Z-axis, representing the horizontal direction. The X-axis is along the vertical direction, and which means XOZ is the rotating plane and ZOY is the horizontal plane. The computing area can be divided into two parts, the inner space of the cuboid and the extended area of the disk [7]. Based on the center of the disk, the extended area has a length of 4D(D is the diameter of the disk) up the stream and 10D down the stream, while the radial diameter is 3D, shown Figure 1. Three blades with the same scale are included in the disk, with a diameter of 70 m. Moreover, the hub height is 65 m, hub radius is 2m, and the angle between main axis of the turbine and horizontal surface is 5°. Leaving the body of the turbine out, the disk now has a cavity inside and is calculated as rotating area with dynamic meshes. The computing model is meshed in GAMBIT, using line mesh method on blades and surface mesh method on all certain faces, as is shown in Figure 2. The disk and the cuboid have 3 interfaces, which are the joints of static and dynamic meshes, so the “interface” function is applied. As the cuboid has a series of bodies inside, and proper mesh quality can both reduce computing time and increase accuracy, this paper divides the cuboid into several parts. For the disk, hybrid-tetrahedral meshes are applied and get 1.79 million meshes in total. While for the other parts, we use hexahedron meshing and get 3.08 million meshes.

![Figure 1. Dynamic mesh model of plane graph](image1)

![Figure 2. Computational grid domain of Dynamic mesh model](image2)

2.1.2. Mesh-preview and computing results of Dynamic mesh model

After the meshing work in GAMBIT, some definitions are in need before the preview [8]. First, define Y-axis as the flow direction, and the inlet is uniform velocity entrance. The outlet is defined as a uniform pressure one. The bottom is treated as solid, smooth wall without slipping. The other surfaces

---

2
are defined as symmetry walls. The flow area is defined as well. Second, import the model into FLUENT15.0 to adjust the parameters, apply “mesh interface” for the joints of static and dynamic meshes, write UFD, and assign the value of rotating speed and time. Third, import UFD profile, and assign the mesh condition together with rotating vector. Have a preview of the moving area and dynamic meshes to make sure the dynamic meshes rotate in the right direction. Last, adopt three-dimensional solver with double precision, along with k-epsilon model. For pressure and velocity coupling, use SIMPLE method. Other assumptions include the intensity of air $\rho$ is 1.225kg/m³, the dynamic viscosity $\mu$ is $1.7894 \times 10^{-5}$Pa·s, the inlet velocity is 12m/s, and the angular velocity of the disk is 17.32r/min.

2.2. Actuator disk model (ADM)
In order to verify the property of using dynamic mesh method and its accuracy, this paper also adopts numerical simulation with actuator disk model.

When air passes through the rotor disk, the velocity within the flow tube is changed, as a result of the resistant effect [9]. According to the law of mass conservation, if we assume the intensity of air keeps constant, the area of the cross-section in flow tube changes contrarily with velocity. That means as velocity goes down, the cross-section becomes larger. As a result, we named the plane of rotor as actuator disk.

From classic Rankine-Froude theory, when the flow passes through the actuator disk, there’s a gap between frontier pressure and rear one, and this gap can be calculated by:

$$\Delta p = 2\rho v_0^2 a(1-a)$$ (1)

$\Delta p$ is the pressure difference when air passes actuator disk, $\rho$ is air intensity, $v_0$ is wind velocity, $a$ is axial induction factor and defined as $1 - \sqrt{1 - C_r^2}/2$. Simplify wind turbines as actuator disks and adopt the function for porous cells:

$$\Delta p = \left( \frac{\mu}{\alpha} v_0 + \frac{1}{2} C_2 v_0^2 \right) \Delta m$$ (2)

$\mu$ is laminar viscosity, $\alpha$ is penetration rate, $C_2$ is pressure coefficient, $v_0$ is normal velocity, $\Delta m$ is the thickness of the medium. Combining Equation (1) and (2), $\alpha$ and $C_2$ is obtained.

3. Simulation results and analysis

3.1. Wake velocity distribution: wake flow velocity contours
In order to come up with a common rule of wake effect, we compare the cross-section of velocity at the distance of D, 2D, 3D, 4D, 5D and 6D(where D is the diameter of the rotor, 70m), for both AD model and dynamic mesh model, as is shown is Figure 3.
We know from Figure 3(a) and Figure 3(b) that the distributions of velocity in the wake flow for both models are quite alike. As the distance after the turbine grows the gradient of velocity becomes smaller, and the influenced area gets larger. This regulation is in good agreement with wake theory. The result of numerical modeling is also used to investigate the average velocity of each section and the thrust factor [10]. In Figure 3(a), the area with a velocity less than 11m/s shows the shape of a circle, more or less. As a result, the influenced area can be measure by radius and the center of the circle. In Figure 3(b), the influenced area in each cross-section resembles the outline of the turbine, and develops spirally along the wake stream. Precisely, the velocity near the turbine sees a little bit increase for the tip part, while shows a decrease for the inner part. Although the influenced area becomes larger as wake developed, this regulation still works. All these show that the distribution of velocity is quite different from the uniform one at entrance.

3.2. *Wake velocity distribution: axial velocity contours*
In order to analysis the distribution of vortex and velocity vector after wind turbine, we generate three cross-sections at the distance of 2D, 6D and 10D (D represents the diameter of rotor), with both AD model and DM model, shown in Figure 5.
From Figure 5(a), we know that the inlet flow forms a round wake area after it passed the actuator disk. The vectors all have the direction towards the center. This is because the outside pressure is higher than inside, so the outside air keeps flowing into the wake area to recover the velocity and kinetic energy. Figure 5(b) shows that, the wake of a single turbine develops in a helix way, and the rotating center is no longer coincident with the symmetrical center. In cross-sections at all distances (in XOZ plane), two vortexes with different velocity always appear, the one with higher velocity happens near the tip and the one with lower velocity happens near the interface of hub center and ground (whose rotating direction is in agreement with that of rotor). As the distance grows, the centers of the two vortexes move downwards; the two centers get closer; the shapes of vortexes also become variable. When air flows through the turbine, its velocity drops dramatically and recovers gradually down the stream, the recovery rate is defined as

\[ \eta = \frac{v_n}{v_0} \]  

(3)

\( v_0 \) is the inlet velocity; \( v_n \) is the average wind speed of the cross-section at the distance of \( nD \). In order to come up with a common rule of developing wake, the velocities of both models are analyzed and compared, shown in Figure 6. Figure 6 shows that, as wake goes down stream, the velocities of the two models both shows some recovery, but at different rates. The computing length (10D) is not long enough for velocity to recover entirely. For distance \( y \leq 4D \), the recovery rates of DM model and AD model are almost the same, and the velocity difference changes quite slow with distance. For distance \( y > 4D \), the recovery rate of AD model are larger than that of DM model, the difference of velocity changes dramatically, with a peak value of 16.77%. That is because when wake flows a certain distance down the stream, the pressure gap is no longer the main factor, turbulence intensity and the variance of vortexes become more important instead. In the true practice, the vortexes will dissipate gradually, leading to a decrease in total energy and velocity.
The above shows that the recovery rate of DM model is smaller than that of AD model and there exists a big disparity. There are mainly two possible reasons. First, in an AD model, it is mass conservation law and momentum conservation law that determine the force on the disk, which will change the local velocity on and near the disk. At the same time, it consumes energy for wake to rotate at certain angular velocity, leading to a static pressure loss. As the wake develops, this loss becomes smaller and that’s why the recovery rate gets larger. While for the true rotor, the force is determined by axial and tangential energy functions. Second, unlike AD model, DM model take the rotating blades into consideration, and this rotation will lead to large scales of circle flow, turbulence and vortex. The dissipation of vortex will cost a lot of energy, resulting in lower velocity. Based on what is stated above, DM model resembles the true practice most, as it considers both rotating effect and precise geometric blade information. In future works, we will also adopt the information of real wind farms to confirm and improve the conclusion.

4. Conclusion
In this paper, we use Fluent15.0 to simulate 1.5MW wind turbines with dynamic meshes. The velocity of wake is plotted and made comparison with that of AD models. Combining with true practice, it is confirmed that DM method has high validity and accuracy. Some conclusions are made as following:

- Build dynamic mesh model, simulate the flow field, get the velocity distribution within the wake of a single turbine. The result can be helpful in investigating the velocity filed in the design process of wind farm.
- Simplify the actuator disk model with theory study and CFD. Use the model to solve the wake flow field, get the distribution of axial velocity and fluid at different distance down the stream.
- With the same wind speed, analysis and compare the wake flow distributions at multiple cross-sections for both two methods. For DM model, two vortexes appear in the down stream and dissipate gradually as distance increases. As for AD model, there’s no such phenomenon. That indicates that dynamic mesh modeling is more precise and convincing, and can capture fluid details in larger scale.
- Comparing the contour plot of velocity at different cross-sections of the two models, we find that the smallest value appears at the distance of 1D after the hub, rather than right after it. As the distance increases, velocity recovers gradually in a linear way for both models.
- Using the recovery rate of velocity, power output, and wind utilizing factor as indicators, there’s a deviation between DM model and AD model. Moreover, as DM model takes the rotating of blades into consideration, its result is quite close to the true practice. In conclusion, DM model is a new method to simulate wind farms, and in terms of reliability and accuracy,
DM method performs better than any other methods. Footnotes should be avoided whenever possible. If required they should be used only for brief notes that do not fit conveniently into the text.

5. References
[1] Sten F, Rebecca B and Sara P 2004 The necessary distance between large wind farms offshore study (Roskilde: Risø National Laboratory) p 2-11.
[2] Barthelemie R J, Rathmann O and Frandsen S T 2007 Modelling and measurements of wakes in large wind farms J. Phys. Conf. Ser. 75 12-49.
[3] Alexandros M and John C 2009 CFD modeling of the wake interactions of two wind turbines on a Gaussian hill (Florence: EACW) p 9-13.
[4] Zhu C, Wang T G and Zhong W 2013 Investigation of wake interaction between wind turbines in tandem J. Mech. Eng. 35 17-22.
[5] Zhou Y L, Yang C Z and Li L 2012 3D Flow field numerical simulation on aerodynamic characteristics of new double-rotor wind turbines J. Chin. Soc. Pow. Eng. 32 698-705.
[6] Li R N, Guo X W and Yang R 2012 The design of 1.5 MW wind turbine blade and its optimization based on the Matlab J. Gans. Sci. 24 73-76.
[7] Li S H, Wang D H and Yue W P 2011 Numerical simulation on wake interaction and array of double wind turbine at varying wind directions J. Chin. Soc. Pow. Eng. 31 768-78.
[8] Li S H, Yue W P and Kuang Q F 2011 Numerical of simulation of wake interaction and array of double wind turbine J. Proc. CSEE 31 101-7.
[9] Tian L L, Zhao N and Zhong W 2012 Numerical simulation of wake interactions of wind turbine J. Acta Ener. Solar. Sin. 33 1315-20.
[10] Qian Y R and Wang T G 2016 Large-Eddy Simulation of Wind Turbine Wake and Aerodynamic Performance with Actuator Line Method J. Trans. Nanj. Univ. Aero. Astro. 33 26-36.
[11] Yuan W, Tian W, Ozbay A and Hu H 2014 An experimental study on the effects of relative rotation direction on the wake interferences among tandem wind turbines J. Sci. Chin. 57 935-49.

Acknowledgement
This work was supported by China Huaneng Group Science and Technology Fund (HNKJ16-H25)