Analysis of aerodynamic characteristics of the wind turbine tower

Yihe Wang¹, Zhen, Liu²*, Yinzhen Liu² and Meiying Liu²

¹ Marketing Departmen, Beijing Urban Construction Yatai Group Co., Ltd, Chaoyang District, Beijing, 100013, China
² College of Energy and Power Engineering, Inner Mongolia University of Technology, Hohhot, Inner Mongolia, 010051, China

*Corresponding author’s e-mail: 20191100157@imut.edu.cn

Abstract. In order to investigate the aerodynamic characteristics of Wind turbine tower under rated operating conditions, a 1:1 numerical model of 1.5MW wind turbine was established using UG finite element modeling software, and the velocity field and pressure field distribution of wind turbine with cross-sections of \( r/R = 20\% \), \( r/R = 40\% \), \( r/R = 60\% \), \( r/R = 80\% \), \( r/R = 120\% \) and \( r/R = 140\% \) were analyzed. The results show that with the increase of \( r/R \), the backflow and vortex on the leeward side of the tower are produced to different degrees, and the vortex separation point is shifted backward. With the increase of \( r/R \), the pressure value and range about 0° symmetry axis gradually become smaller, and the pressure values on both sides of the tower show negative pressure and symmetric distribution, and the tail flow area of the tower gradually returns to the inlet pressure.

1. Introduction

The tower, as an important part of supporting the wind turbine, bears its own weight and that of the wind turbine, the load of the wind turbine, and the wind flow. Therefore, the influence of the tower shape on the flow field is particularly important. Domestic and international scholars have conducted explorations: Braham et al. [1] investigated the effect of airflow structure generated by nacelle and tower on the near wake of 2.5mw wind turbine. The study by Bobonea et al. [2] tried to analyze the flow field around the wind turbine and emphasized the effect of a constant blowing wind at the leading edge of each blade to improve the performance of the wind turbine. Gan et al. [3] modeled the tower of the lower wind turbine presented the tower shadow profile, and analyzed the tower shadow characteristics for both the cylindrical and lattice tower types. Sintra et al. [4] studied the effect of cylindrical towers on the wake flow and output power of wind turbines. Liu Haifeng et al. [5] used numerical simulation to analyze the aerodynamic characteristics of two different types of towers. In this paper, based on these studies, the pressure and velocity distributions of a 1.5 MW wind turbine at different cross-sections are analyzed.
2. Calculation model and calculation method

2.1. Model and computational domain creation
The 1.5MW wind turbine is modeled 1:1, as shown in the wind turbine model diagram established in Figure 1. The wind turbine diameter is 78m, tower height is 67m, hub height is 68m, hub diameter is 1.5m, and rated speed is 16.85.

![Wind Turbine](a) Wind Turbine  ![Wind Turbine assembly](b) Wind turbine assembly

Figure 1. Wind turbine model.

2.2. Calculation method
Detached-Eddy Simulation is able to simulate large separated flow better than the RANS method. Compared with Large-eddy simulation, it is able to capture many nonstationary states that the RANS method is unable to do when simulating the same large separated flow, overcoming the huge computational overhead problem caused by solving for all turbulence scales, while the number of meshes is much lower than that of the LES method. When the case of a denser mesh near the wall surface occurs, this performs the LES and reduces the eddy viscosity (i.e., model stress), so the DES method is selected for this study.

2.3. Computational domain and meshing

2.3.1. Calculation domain size. Figure 2 shows the CFD model and the structure of the flow field. The computational domain is divided into two parts, the rotating domain, and the stationary domain, and the diameter of the rotating domain is 68m, whose value is the diameter of the wind turbine. In the stationary domain, the length of the inlet flow field is 1D, and the wakefield (outlet flow field) is 15D, which is verified to be in accordance with the actual situation.

![CFD model](Figure 2. CFD model.)

2.3.2. Dividing the mesh. The success or failure of the numerical simulation is determined by the merits of the mesh division, and a better mesh division can achieve optimized results in terms of computational time, accuracy, and stability and then solve the computational problem. The mesh distribution is shown in Figure 3.
The minimum encrypted block length and width are 2D, 1D, and 1D, and the maximum encrypted block length and width are 5D, 1.5D, and 2D, respectively. The smaller denser area size is 0.6m, and the minimum size of the larger encrypted area is 2.64m. The total number of grids in the wind turbine and the computational domain is 6.6 million.

In order to get a more accurate calculation, the size of the blocking rate $Q$ is used to indicate the number of computational domain grids accurately. The blocking rate equation is as follows:

$$Q = \frac{A_o}{A_1} \times 100\%$$

(1)

Where $A_o$ is the obstacle windward area $m^2$, $A_1$ is the calculation domain cross-sectional area $m^2$, the calculation domain cross-sectional size is $4.5D \times 6D \times 16D$, the obstacle windward area $0.78 \, m^2$. This time the blockage rate is $Q = 0.28\%$, the blockage rate is less than 3%, the size of the grid size layout is more reasonable. In order to make the simulation calculation closer to the experimental situation, the final total number of grids is about 6.6 million.

2.3.3. Boundary Conditions. The computational domain inlet is defined as the velocity inlet direction with a value of 11.02, and the computational domain outlet direction is defined as the pressure outlet direction with a relative pressure of 0 Pa. The blade, hub, motor, tower, and rudder all adopt the boundary conditions without slip.

3. Analysis of the pneumatic characteristics of the tower under rated working conditions

3.1. Velocity field distribution under different sections of the tower

The rated wind speed is $v = 11.02 \, m/s$, and the six cross-sections distributed in the $y$-axis direction are $r / R = 20\%$, $r / R = 40\%$, $r / R = 60\%$, $r / R = 80\%$, $r / R = 120\%$ and $r / R = 140\%$ sections, where $r$ is the distance from the wind turbine rudder to the selected section in the $y$-axis direction and $R$ is the wind turbine radius ($R = 39m$).
As shown in Figure 4, only the tower in the flow field acts in the wind tunnel, and the windward side is in the low-speed region in the high-speed region on both sides of the tower, the windward side separation point location on different cross-sections is basically the same. With the increase of $r/R$, the windward area and the low-speed area of the wakefield become narrower from wide, and the wake area gradually returns to the inlet wind speed. The leeward side of the tower produces different degrees of backflow and vortex, and a more symmetrical vortex shedding appears in the wake area on the same cross-section, and the vortex separation point is shifted backward as the $r/R$ becomes larger.

3.2. Pressure distribution under different sections of the tower

Figure 5 shows the pressure coefficient distribution of six cross-sections distributed in the wind turbine blade spreading direction. The pressure reaches its maximum value from $\theta=0^\circ$, and then starts to decrease along the circumference of the tower, reaching its minimum value at around $\theta=90^\circ$, and the pressure gradually rises. The pressure becomes stable when $\theta=160^\circ$, at which time the tower shear stress tends to 0, until $\theta=232^\circ$ the pressure once again starts to drop to $\theta=270^\circ$ to reach the minimum value. After that, the pressure starts to rise again, showing a w-shape, with a period of $0^\circ$-360° and a more symmetrical pressure distribution. The simulated tower pressure extremum curve is consistent with the experimental results of West and Apelt[6]. With the increase of $r/R$, the pressure value gradually becomes smaller.

![Figure 4. Velocity streamline diagram of different cross-sections of tower.](image)

![Figure 5. The pressure coefficient of the tower under different cross-sections.](image)
4. Conclusion

(1) With the increase of $r/R$, the windward area and the low-speed area of the wakefield become narrower from wide, and the wake area gradually returns to the inlet wind speed. The leeward side of the tower produces different degrees of backflow and vortex, and a more symmetrical vortex shedding appears in the wake region in the same cross section, and the vortex separation point shifts backward as the $r/R$ gets larger.

(2) With the increase of $r/R$, the pressure value and range about 0° symmetry axis gradually become smaller. It is caused by the increasing of $r/R$ and the changing of tower diameter profile. The pressure values on both sides of the tower show negative pressure, are symmetrically distributed, and the wake area of the tower gradually returns to the inlet pressure.

Acknowledgments

The authors wish to acknowledge the support and motivation provided by the Key Projects in the horizontal scientific research project: Monitoring of flow field characteristics around wind turbine tower.

Reference

[1] Abraham, A., Dasari, T., Hong, J. (2019) Effect of turbine nacelle and tower on the near wake of a utility-scale wind turbine. J. Journal of Wind Engineering and Industrial Aerodynamics., 193: 1-11.

[2] Bobonea, A., Berbente, C., Niculescu, M. L. (2014) Horizontal axis wind turbine flow field analysis and prospects for performance enhancement. J. Upb Scientific Bulletin., 76: 45-54.

[3] Gan, L. K., Shek, J., Mueller M. (2017) Modeling and Characterization of Downwind Tower Shadow Effects using a Wind Turbine Emulator. J. IEEE Transactions on Industrial Electronics., 64: 7087-7097.

[4] Sintra, H., Mendes, V. M. F. (2014) Modeling and simulation of wind shear and tower shadow on wind turbines[J]. Technology Procedia., 17: 471—477.

[5] Liu, H., Zhu, B., Zhang, H., et al. (2019) Comparative Study on Wake Disturbance Characteristics of Wind Turbine Generator with Steel Tube Lattice and Cylinder Tower. J. Acta Energetica Sinica., 40: 2036-2044.

[6] Xu, C., Chen, L., Lu, X. (2012) Large-eddy simulation of the compressible flow past a tabbed cylinder. J. Chinese Science Bulletin., 665: 238-273.