Studies on the Influence of Building Spoiler on Performance of Distributed Vertical Axis Wind Turbine

Chi Zhang¹, Yan Jia¹, Meng Zhao¹, a, Wenxiong Li¹, Yao Wang¹

¹College of Energy and Power Engineering, Inner Mongolia University of Technology, China

a mzhao_imut@163.com

Keywords: Vertical axis wind turbine; CFD; wall spoiler; sliding mesh

Abstract. As the trend of novel energy development, distributed power generation can take fully various decentralized energy sources. However, the buildings around the installation site may greatly impact the wind turbines’ performance. Herein, a novel type of combined vertical axis wind turbine is investigated with a sliding mesh technique in terms of how the upstream wall disturbs incoming winds at different speeds, the influence area of turbulent vortex, and the performance variation. The results show that the building spoiler will affect the speed and flow direction of the wind horizontally and vertically in the downstream areas. Wall’s influence on wind speed fluctuation in downstream area is systematically analyzed. It turns out that under the premise of scarcely affecting the performance of wind turbines, disturbing flow area or low disturbing flow area should try to be avoided when choosing the installation position.

Introduction

Vertical axis wind turbines have been widely used in distributed power generation. However, limitations for installation site can let the upstream building disturbance somehow impact the downstream space with respect to the speed, direction, and turbulence intensity of the wind field, and the aerodynamic performance of downstream wind turbines may be further influenced when these parameters are varied. Therefore, it is necessary to clarify how the building turbulence impacts the performance of downstream wind turbines.

Early foreign scholars simulated and figured out the aerodynamic performance of vertical axis wind turbine on the basis of a two-dimensional flow field, and the applicability of sliding mesh was verified in the calculation of vertical axis wind turbine blade technology [1-3]. Domestic scholars have studied the performance of H-type and S-type vertical axis fans in terms of wind speed and turbulence, and further analyzed roof wind flow direction so as to alter the wind farm environment at downstream and control the operating efficiency of downwind wind turbines [4-8]. In the present study, a novel type of combined vertical axis wind turbine with Φ-type outer blades as lift blades and S-type inner blades as resistance blades are taken as the research object. The influence of wall disturbance on the performance of vertical axis wind turbine generator is studied by changing the wind speed and simulating the working conditions under both wall-existing and wallless conditions.

Numerical simulation method

Turbulence model. Compared with the standard k-ε model, RNG model that considers the turbulent vortices can guarantee favorable accuracy and be better applied in bending, swirling, and rotating strong streamlines [9].

Geometric Modeling and computational domain mesh generation

Wind turbine parameters. Only the wind wheel part of wind turbine is focused in this research. The parameters involved in wind turbine and wall are as follows:
Table 1. The parameters involved in wind turbine and wall

<table>
<thead>
<tr>
<th>Calculation parameters</th>
<th>Parameter data</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wind turbine type and rated power [W]</td>
<td>700</td>
</tr>
<tr>
<td>Rated wind speed [m/s] / rated speed of wind turbine [rpm]</td>
<td>12/415</td>
</tr>
<tr>
<td>Blade height of wind turbine internal resistance [m]</td>
<td>0.446</td>
</tr>
<tr>
<td>Part height / sweep diameter of wind wheel [m]</td>
<td>1.547 / 1.92</td>
</tr>
<tr>
<td>Cut-out speed [m/s]</td>
<td>25</td>
</tr>
<tr>
<td>Endurance wind speed limit [m/s]</td>
<td>60</td>
</tr>
<tr>
<td>Wall height [m] / thickness [m]</td>
<td>3 / 0.5</td>
</tr>
<tr>
<td>Distance between wall and wind turbine [m]</td>
<td>4</td>
</tr>
</tbody>
</table>

Fig. 1, Schematic diagram of fan size and position coordinates

**Calculation domain creation and meshing.** The method of controlling blocking rate Q is adopted in the computation domain. Herein, an area of 8D×4D×4D (D represents the sweeping diameter of wind turbines) is selected as the calculation domain. The blocking rate is between 2.5% and 7% according to the dimension of the fan mentioned above, which meets the requirement in the whole calculation area[^10].

**Physical model and boundary condition setting.** Three dimensional models, hidden transient solutions, separation solvers, gas and N-S equations are selected. The rotation rate is set as n=415rpm in the RNG k-ε turbulence model, and the vertical axis wind turbine is numerically simulated at five different wind speeds (v₁=6m/s, v₂=9m/s, v₃=12m/s, v₄=15m/s, v₅=18m/s). The computation domain consists of wind speed as the inlet and pressure as the outlet, with a relative pressure of 0Pa.

**Numerical results and analysis**

**Numerical simulation results under wallless condition.** The wind power and power coefficient are extracted from the simulation results under wallless condition.

Fig. 2, Variation curves of power and power coefficient with incoming wind speed
Figure 2 depicts the trend of power curve from numerical simulation is basically consistent with that provided by the fan manufacturer, which further verifies the simulation accuracy.

**Numerical simulation and results under wall-existing condition.**

![Figure 3](image)

Fig. 3, Wind turbine speed distribution diagram under wall-existing conditions

When wall exists, at different inflow wind speeds, Figure 3 depicts that as the incoming air flows through the wall, there is a certain region of swirling turbulent in the upper and lower parts of the fan, and the wall-generated tangential wind velocity affects the fan performance significantly. In order to study the influence range of turbulent region on the fan, the monitoring points are set within certain ratio area, including the ratio of specific horizontal distance to wind turbine radius (L/R=-6,..., 14, scale 1) and the ratio of vertical distance to fan height (H/h=-3,..., 3, scale 0.5). The average maximum wind speed at the monitoring point is plotted against the incoming wind speed, and the change curve of the velocity ratio (the ratio of the maximum speed to the incoming wind speed, $v/v_0$) is plotted as well.
Fig. 4, Variation curves of wind speed with incoming wind speed under wall-existing condition

Fig. 4 suggests that the tendency of how wind speed varies with the incoming wind speed is the same for both wall-existing and wallless condition. But since there is a wall 4m in front of the fan, the wind on the horizontal axis is turbulent in the upstream and downstream areas of the fan—the higher the incoming wind speed, the greater the change of wind speed. The wind speed in the 2R–13R interval exceeds that of the wind turbine cut-off, i.e. 25m/s, and the wind turbine stops generating electricity in this range. After reaching 18m/s, the wind speed in the range of 6R–10R exceeds the wind speed limit (60m/s), and the structure of the wind turbine will be destroyed in this range. When the wind speed ratio at the 14R position approaches 1, the area that the wall turbulence can affect the fan downstream is also around 14R. Graphs (c) and (d) plot the wind speed along the vertical axis versus the incoming wind speed. The lower edge of the fan (at about 0.5H of height) due to fan rotation will disturb the wind speed here to a certain extent. The curve shows that the wind speed ratio in the height range of 0–1H is relatively steady and close to 1, indicating that the extent of the wall disturbance is less affected. The position at height of 1.5H is affected by wall interference the most. In summary, the installation position of wind turbine should keep away from the spoiler area or low disturbance area as far as possible.

Comparison and analysis of simulation results with wall-existing and wallless condition. In order to analyze the influence of wall spoiler on wind speed at wind turbine, the average wind speed is monitored at the upper edge, lower edge, center, and windward blade of the wind turbine under two different conditions. The wind speed ratio (the ratio of wind speed under wall-existing condition to that under the wallless condition, \( \frac{v_w}{v_0} \)) is calculated and plotted against flow wind speed curve.

Fig. 5, Variation curves of fan position velocity ratio with incoming wind speed

Figure 5 shows that the wind speed at fan position under wall-existing condition is higher than that under wallless condition when the incoming wind speed is below 12m/s, and the wind speeds at both fan center and blade position under wall-existing condition are lower than those under wallless condition when the incoming wind speed is above 12m/s. The swirling turbulence of the wall impacts the lower edge of the fan to the most while the upper edge to the least. Combining the velocity distribution diagram of the wall in Fig. 3, this phenomenon can be explained as follows. At wind speed below 12m/s, the vortex turbulent region of wall is not separated completely from the fan lower
edge of and the leeward blade position. The wind speed in the vortex turbulent region is higher than
the incoming value, and some turbulent regions will raise the wind speed at the fan position.

According to the calculation results, the power variation along with the wind energy utilization
coefficient under the two working conditions are plotted.

![Variation curves of power and Power coefficient](image)

Fig. 6, variation curves of power and Power coefficient

The power and wind energy utilization coefficient curves in Fig. 6(a) show that the power
increases with increasing wind speed in both cases. Under the wall-existing condition, the power
increases gently at wind speed greater than 12m/s while becomes higher than that under the wallless
condition at incoming speed below 12m/s. This is because when the incoming wind speed is lower
than 12m/s, the wind speed at the windward blade and the center position of the fan is higher under the
wall condition, thus the increased torque will result in a power increase. The growth rate is relatively
large at incoming wind speed of 6m/s, which generates approximately a fivefold increase in power
compared with the wallless state. But at incoming wind speed higher than 12m/s, the power under the
wall-existing condition is lower than that under the wallless condition, and it even decreases by
43.36% at the velocity of 18m/s. Fig. 6(b) also suggests a decreasing trend of the wind energy
utilization coefficient with increasing wind speed under wall-existing condition. Compared with the
wallless working condition, the wind energy utilization coefficient under the wall working condition
is higher at wind speed lower than 12m/s, and its growth rate is relatively large at is 6m/s, about 5.46
times higher than that of the wallless working condition. Contrarily, the wind energy utilization
coefficient under wall conditions becomes lower than that under the wallless working condition when
the wind speed is higher than 12m/s. The drop rate is about 44.01% when the incoming wind speed is
18m/s.

**Conclusions**

1. Verifies the accuracy and applicability of the Star ccm+ software for the simulation analysis of
   the fan performance.
2. The building spoiler will affect the speed and flow direction of the wind horizontally and
   vertically in the downstream areas. This causes the wind speed in the downstream part to exceed the
cut-out speed and endurance wind speed limit of the wind turbine.
3. At low wind speed, the wind turbine is located in the vortex area of wall turbulent, the wind
   speed at which place is higher than the incoming speed; this results in an increased wind power and
   higher utilization coefficient of wind energy. On the contrary, for high wind speed, the wind turbine is
   separated from the turbulent vortex, which leads to a lower wind speed at fan position than the
   incoming speed along with reduced power and energy utilization coefficient.
4. Wall’s influence on wind speed fluctuation in downstream area is systematically analyzed. It
   turns out that under the premise of scarcely affecting the performance of wind turbines, disturbing
   flow area or low disturbing flow area should try to be avoided when cho’osing the installation position.
Acknowledgements

This work was financially supported by the Study on wind/solar/biomass hybrid power system with stabilization storage system (201601064), Research on the aerodynamic load of high-speed train pantograph under crosswind(2018LH01011), Study on the aerodynamic characteristics about high-speed train pantograph under cross wind(NJZY17105) and Analysis and research on aerodynamic characteristics of pantograph for high speed Train under strong cross wind(ZD201706).

References


