6th BSME International Conference on Thermal Engineering (ICTE 2014)

Adjacent wake effect of a vertical axis wind turbine

Harun Chowdhury*, Israt Mustary, Bavin Loganathan and Firoz Alam

School of Aerospace, Mechanical and Manufacturing Engineering, RMIT University, Melbourne, 3083, Australia

Abstract

The main objective of this study is to understand the effect of turbine placement and surrounding structures. Using Urban Green Energy’s UGE-4K vertical axis wind turbine and the ANSYS computational fluid dynamics package (CFX), a dynamic fluid analysis was undertaken looking at the wake of the turbine through a variety of different inlet speeds and rotational frequencies to determine suitable flow recovery for optimal placement of subsequent turbines. The results showed that the wake interference is minimal at around 5 times the diameter of the turbine downstream. Results also show that flow recovery was a lot slower to the right of the turbine especially along a line 15° from the centre of the turbine to the right as this is coincident with the vortices generated from the turbines rotation.

* Corresponding author. Tel.: +61 3 99256103; fax: +61 3 99256108.
E-mail address: harun.chowdhury@rmit.edu.au

1. Introduction

Harnessing the wind for power has been achieved by humans for over 5000 years with the Egyptians using sails to travel the Nile. The first windmills were discovered around the Persian-Afghan borders date back to 200 BC and the Dutch followed with their windmills for irrigation and drainage around 1300-1875 AD [1]. When the electrical generator was invented the application was soon used with the wind turbine and the first large wind turbine to
generate electricity was created in 1888 and was rated at 12 kW [2]. The progression continued with technology levels increasing and more research being undertaken it wasn’t long till large scale wind farming was taking place with a notable case of California where over 16,000 machines ranging from 20-350 kW were installed between 1981 and 1990 for a total of 1.7 GW [3]. In recent years the global wind industry has been the fastest growing industry in electrical power generation with over 237,000 MW worth of capacity installed by the end of 2011.

The vertical axis wind turbine (VAWT) has been around for years but is starting to get more and more widely used as the world constantly increases its focus on renewable energy sources. The VAWT has the capability of providing a renewable energy source to the domestic, private and small commercial markets as it can be placed on rooftops and areas of turbulent flows with more success than a horizontal axis wind turbine (HAWT) [4-6]. The wind turbine power generation efficiency in commercial and domestic applications can significantly be affected by the built up geometry and the interference of adjacent turbines [7]. In order to maximize the power output, it is important to understand aerodynamic behavior of different building edges, surrounding structures and nearby installed turbine wakes. The wake of a turbine is an important part of analysis because it gives an assessment of the performance of the wind turbine, the other benefit of wake analysis is that an understanding of the downstream effect of the flow can be garnered and used for the optimizations of the placement of subsequent wind turbines for maximum effectiveness. Despite the importance, little information is available on optimal placement of turbines and their wake effect. Hence, the main objective of this study is to understand the effect of turbine placement and surrounding structures. Numerical and computational fluid dynamics (CFD) models were used to analyze wake and turbulence in the analysis of a commercially manufactured VAWT.

2. Methodology

2.1. Selection of wind turbine

Urban Green Energy (UGE) is a world leader in small wind and renewable energy systems, with installations across the globe. UGE designs, manufactures, and markets cutting-edge vertical axis wind turbines and hybrid renewable solutions with a track record of high performance, safety, and reliability. Fig. 1(a) shows the UGE-4K turbine marketed towards large domestic or moderate commercial scale. Detailed specifications of this wind turbine can be found in [8]. A simplified 3D CAD model of the UGE-4K turbine was developed for this study as shown in Fig. 1(b). The simplified CAD model of the turbine was used as the downstream wake profile is dependent on the whole physical geometry and rotational speed of the turbine rather than aerofoil type blade profile.
2.2. Analysis parameters

In order to get a good understanding of the wake effects, the turbine was analysed across a range of expected wind speed and at a range of associated rotor speed. From the UGE-4K data sheet obtained from UGE, the cut-in speed was found 3.5 m/s and its optimal running speed was 125 rpm at 12 m/s wind speed. Therefore, a good range for analysis would be 2–12 m/s to look at the flow around a turbine before cut-in to investigate the flow behavior of subsequent turbine for cut-in, and not above 12 m/s because there is a decrease in efficiency above the optimal speed and the expected local wind speeds in the built-up environment would rarely be above a clean 12 m/s flow.

2.3. CFD setup

The CFD package utilized was ANSYS CFX for its ability to model and analyse rotating turbines with the help of its TURBO model setup. The turbine was boolened out of a cylinder with 0.25 m clearance on all sides of the blades. This cylinder was then fitted inside the fluid domain with all edges matching up with a negative space of the same dimensions so that there would be no issues with surface interfaces. The cylinder acts as the rotating domain immersed in the fluid domain. The model was analyzed in full-scale of the UGE-4K; thus the rotating domain has 3.5 m diameter and 4.9 m height. The fluid domain dimensions are 66 m × 45 m × 30 m. The model was meshed with more importance placed on the fluid domain as it is the point of analysis rather than near boundary layer of the blades of the turbine. Thus the quality of the mesh is aimed at giving reasonable results within the fluid domain. A tetrahedral mesh was used with refinement on curvature and proximity to geometry. Mesh statistics indicated an average skewness of 0.232 which is an acceptable value for the purposes of this analysis as it is expected to yield reasonably accurate results. Fig. 2(a) shows the position of the rotational domain inside of fluid domain whereas mesh of the fluid and rotational domain are shown in Fig. 2(b) and Fig. 2(c) respectively.

![Fig. 2. (a) rotational domain modelled inside of fluid domain; (b) fluid domain mesh; (c) rotational domain mesh.](image)

The fluid used in this analysis is standard air at ground level, i.e. at 25 °C with a density of $\rho = 1.185 \text{ kg/m}^3$ and a dynamic viscosity of $\mu = 1.831 \times 10^{-5} \text{ kg/m s}$. As the turbine was analyzed at a range of speeds the inlet fluid velocity from 2–12 m/s. The ground was treated as a wall with no slip conditions. The sides and sky are defined as openings with a zero relative pressure and zero relative temperature difference to that of the fluid body. The outlet is treated as an outlet with an average static pressure of zero. The rotating domain was set as a rotating body with input revolutions across 24–125 rpm with respect to the inlet speeds and was rotating clockwise as the turbine is designed to do. The blades and body of the turbine are considered as rotating non-slip walls. There are interfaces defined between the matching faces of the rotating domain and the fluid domain. The interfaces are placed as rotor-stator with a pitch change of zero as the whole model is present and rotating 360 degrees. The $k-c$ turbulence model was used for the analysis. The solver control was set so that it runs across the physical timescale under which the turbine completes one full rotation. The residual type was defined as RMS rather than Max Residual which leads to lose convergence, the iterations to convergence was defined as 100 and the convergence criteria was $10^{-4}$. 
3. Results and discussion

Usually two turbulent models: $k-\varepsilon$ and $k-\omega$ are widely used for the analysis of CFD simulation [9]. The most appropriate turbulence model for this analysis was determined through running two simulations using the both turbulent models on a sphere under similar conditions by matching the inlet wind speed with respect to the equivalent Reynolds number of the VAWT simulation. Table 1 shows the results from the simulations. Using the resulting calculated $C_D$ on the sphere for both turbulence models and comparing them with the graph of $C_D$ vs. $Re$ on a sphere to determine which was closer to the expected for the relative turbulent region under which the model would be evaluated. With the expected drag on a smooth sphere around the region with a Reynolds number of $1.96 \times 10^6$, is around 1.5-2.0 [10]. It can be seen in Table 1 that the $k-\varepsilon$ model yields a closer $C_D$ value for this turbulence region with 0.14729 and thus this model is more suitable for the wind turbine analysis in this turbulence region.

Table 1. Results of CFD simulations using $k-\varepsilon$ and $k-\omega$ models on a sphere.

<table>
<thead>
<tr>
<th>Parameters</th>
<th>$k-\varepsilon$</th>
<th>$k-\omega$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frontal area of sphere (m²)</td>
<td>0.3927</td>
<td>0.3927</td>
</tr>
<tr>
<td>Drag force on sphere (N)</td>
<td>30.843</td>
<td>21.996</td>
</tr>
<tr>
<td>Coefficient of drag of sphere ($C_D$)</td>
<td>0.14729</td>
<td>0.1050</td>
</tr>
</tbody>
</table>

For the wake analysis the results of the range of inlet wind speeds and relative rotor speeds were correlated and compared to each other to determine a value of recovery across the different conditions along different paths within the fluid domain. Velocity and pressure contours, velocity vectors and streams are evaluated for the 2-10 m/s inlet speeds at 24-120 rpm, these contours are based on a plane intersecting the turbine and consequently fluid domain in the centre, thus these results are about the mid-plane. It was observed that there is no real difference between the different inlet speeds and wake patterns but a scale effect, this is because they are in the same turbulence region as the Reynolds number doesn’t vary so much between an inlet speed of 2 m/s to that of 12 m/s.

Fig. 3 shows the velocity contour about the vertical mid-plane and the horizontal mid-plane at 10 m/s inlet wind speed and 120 rpm rotor speed. It can be seen that there is an increased velocity coming off of the top and bottom of the turbine as a result of the flow over the vortexes created by its rotation and low pressure region. There is also an increased total velocity coming off the centre back of the turbine which gradually slows down and returns to around the inlet velocity. The slow regions of velocity are most critical around the front of the turbine and coming off the top and bottom created by the blades creating a low and high pressure region through extracting energy from the wind.

![Fig. 3. velocity contours: (a) about the vertical mid-plane; (b) about the horizontal mid-plane.](image)

Fig. 4 shows the pressure contour about the vertical mid-plane and the horizontal mid-plane at 10 m/s inlet wind speed and 120 rpm rotor speed. The figure shows that maximum pressure occurs in the centre immediately in front of the turbine with low pressure regions above and below the blades with a maximum point towards the back of the turbine. The low pressure regions slowly recover pressure back to zero relative pressure to the inlet downstream with...
the centre of the turbine having the quickest recovery as it is out of the way of the critical low pressure regions at the top and bottom of the turbine. These low pressure regions would also be a critical component of the increased flow over the top and bottom of the turbine in terms of wind speed.

The velocity vector plot is shown in Fig. 5 where only the free stream velocity vector which is aligned with the inlet flow and it gives a more accurate representation of flow recovery because the flow running parallel to the inlet flow is the most desirable flow for the turbine to extract energy from the wind. The plot shows decreased flow immediately around the turbine which gradually recovers downstream and is around 3.5 m/s, increased flow above and below the slow flow of the turbine, at a maximum of around 12.5 m/s, which fans out and slows down back to inlet speed gradually.

![Fig. 4. pressure contours: (a) about the vertical mid-plane; (b) about the horizontal mid-plane.](image)

![Fig. 5. velocity vector: (a) about the vertical mid-plane; (b) about the horizontal mid-plane.](image)

The streamline plot shows the vortices coming off the right hand side of the turbine at the top and bottom and continuing to swirl all the way downstream at an angle of roughly 15° to the centre of the turbine (see Fig. 6). These vortices would indicate that there would be more turbulent flows on the right of the turbine and thus slower recovery on that side. The side on which the vortices are coming off is determined by the direction of rotation of the turbine. If the turbine was rotating counter-clockwise the vortices would be on the left and the reduced flow recovery, in the directional vector, would be slower on the left.

Fig. 7 shows the velocity vector directly downstream from the centre of the turbine for the various inlet speeds and rotor speeds. The results show that better the flow recovery is possible at further downstream. The graph shows that recovery takes a slightly longer distance as the inlet speed increase but for all inlet speeds it can be seen that at around 15 m downstream, and to non-dimensionalise it, divide it by the turbine diameter (3 m) which equates to 5 times the diameter of the turbine downstream, there is almost complete flow recovery. The graph also shows that the flow recovery directly downstream appears to be asymptotic to the relative inlet speed.
4. Conclusions

From the wake analysis over a range of expected wind and rotor speeds it was shown that directly downstream of a turbine adequate flow recovery was seen 5 times the diameter of the turbine downstream. It was also seen that flow recovery was a lot slower to the right of the turbine especially along a line 15° from the centre of the turbine to the right as this is coincident with the vortices generated from the turbines rotation. The converse of this was found in that flow recovery to the left of the turbine was a lot quicker than the right side and central line and that at −15°.

References