Experimental and Numerical Investigation of the Effect of Rotor Blockage on Wake Expansion

B. Akay  
b.akay@tudelft.nl  
DUWIND, Delft University of Technology, Kluyverweg 1, 2629HS Delft, The Netherlands.

C. Simão Ferreira  
C.J.SimaoFerreira@tudelft.nl  

G. van Bussel  
G.J.W.vanBussel@tudelft.nl  

G. Tescione  
Giuseppe.Tescione@gmail.com  
Università degli Studi di Roma "La Sapienza", Rome, Italy.

Abstract

A detailed quantitative description of the aerodynamics of a horizontal axis wind turbine (HAWT) is difficult due to complexity of the flow field. Several methods from experimental to analytical are used to investigate the aerodynamics of a HAWT. In the present study, a wind tunnel experiment and computational fluid dynamics (CFD) simulations are used to explore the expansion of the wake. 2D actuator disc (AD) simulations are compared with the wind tunnel experiments. To understand the aerodynamic behavior of a model wind turbine blade, a detail flow field measurements in chordwise-spanwise directions and in the wake have been done. The measurements are performed on a 2 bladed rotor by means of Stereo Particle Image Velocimetry (Stereo PIV) in an open jet wind tunnel. In this paper, the velocity measurements performed in the wake region of the blade is presented. Actuator disc simulations are performed by applying a constant pressure jump on a permeable disc of zero thickness. Actuator disc simulations are carried out by using FLUENT 6.3.26 with the incompressible version of the Reynolds Averaged Navier-Stokes (RANS) equations. By validating the simulations with the experimental results, one may conclude that the unsteady CFD modeling works correctly and the wake expansion of the prescribed model is affected by the geometry of the Open Jet Facility (OJF).

1 Introduction

To be able to carry out a detailed experimental investigation of a turbine’s aerodynamic behavior, the following set of measurement data would be ideally required: surface pressure measurement, measurement of the 3D inflow distribution in the near wake and the rotor plane, measurements of the wake geometry to establish the expansion of the wake and location of the tip vortices [5]. Wind tunnel experiments must be analyzed by understanding three points; (a) wind tunnel experiments are limited by issues of scaling (b) experiments in open jet facilities lead to expansion of the wake that might differ from a freestream incoming flow field case (c) the limitations of measurements in an open jet wind tunnel.

Glaucert [2] analyzed the wind tunnel blockage in connection with experimental test of propellers in the case of a constantly loaded rotor disc in a closed test section tunnel. Mikkelsen and Sørensen [4] improved the generalized actuator disc method which is modified to cope with the influence of tunnel walls by reducing the set of equations for the induced wind speed through the rotor. Various wake states have been analyzed by Sørensen et al [7] by combining the actuator disc principle with the Navier-Stokes equations. Besides wall correction models for rotors in wind tunnels with closed test sections, Sørensen et al. [6] has performed a study for the wind tunnels with open test section. This study has shown that a simple momentum analysis is not enough to determine the flux of axial momentum from the surroundings into the wind tunnel. Therefore, detailed velocity measurements or CFD predictions are needed. This work is thus aiming to investigate the expansion of the wake of the rotor used in the wind tunnel experiments to see if it is affected by the geometry of the wind tunnel by means of Stereo PIV and CFD. Stereo PIV is capable of accomplishing quantitative measurements of three components of instantaneous velocity vectors.
In this work, the flow field is modeled by CFD for freestream and wind tunnel cases and compared with the experimental measurements carried out OJF. Also the experimental data is used to validate the CFD modeling. Two questions are considered in the present study: (1) Is the CFD modeling valid? (2) Is the wake expansion independent from the geometry of the OJF? To answer these questions two different solution methodologies are followed. First, the wind tunnel experimental setup and conditions are simulated by CFD and simulations are validated with experiments. Second, operation of the rotor in free stream conditions is simulated by CFD and compared with both the experimental results and the simulation in wind tunnel conditions.

2 Experimental Investigation

The experimental work has been performed at the new Open Jet Facility-OJF (see Figure 1) of the TU Delft by using Stereo Particle Image Velocimetry to study the aerodynamics of horizontal axis wind turbine. The maximum test section velocity is \(30 \text{ m/s}\) and the octagonal jet has an equivalent diameter of \(3 \text{ m}\). There are fine mesh screens to reduce the turbulence and velocity deviations in the airflow. The experiment was performed on a 2 bladed rotor which has \(1 \text{ m}\) radius.

![Figure 1: Schematic representation of OJF](image)

From chordwise measurement set-up (see Figure 3), 39 measurement planes are investigated at the tip. The field of view was set in the \(xy\) plane by the use of the traverse system in both axes to track the tip vortex and its evolution. The first window was centered at the blade tip and caught the vortex release and its initial evolution. Five measurement planes are investigated for this analysis; up to \(0.4R\) downwind and \(0.265R\) outboard (see Figure 4). For the whole test a total of 40 sets of 25 images were recorded. Measurement conditions and parameters are tabulated in Table 1. Velocity vectors are obtained using the DAVIS software from 25 couples of images for tracing the tip vortices. With multi-pass refinement and 75% window overlapping, the final vector spacing is \(0.0092x0.0092\) (non-dimensionalized by the tip chord, \(c_{tip} = 0.0643\text{m}\)).

![Figure 2: HAWT model in OJF](image)

3 Computational Investigation

Computational domains and computations are performed by using GAMBIT 2.4.6 and FLUENT
6.3.26 respectively.

A numerical solution to the axisymmetric actuator disk problem has been performed. Computations have been performed with the incompressible version of the Reynolds Averaged Navier-Stokes (RANS) equations and the SST $k\omega$ turbulence model. The actuator disc model corresponds to the size of the test rotor which has 1 m radius.

### Table 1: Measurement conditions and parameters

<table>
<thead>
<tr>
<th>TSR</th>
<th>RPM</th>
<th>$U_{\infty}$ [m/s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>720</td>
<td>10.8</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>FOV [pixels]</th>
<th>FOV [mm]</th>
<th>$\delta$/T [ms]</th>
</tr>
</thead>
<tbody>
<tr>
<td>4872 x 3248</td>
<td>307 x 142</td>
<td>60</td>
</tr>
</tbody>
</table>

### 3.1 2D Computations

The disc is located 8.5\(R\) downstream of the inlet and the domain is extended 20\(R\) in the behind of the disc and 5\(R\) above the disc. The computational domain for the freestream case is shown in Figure 5. The mesh is composed of 24000 structured cells clustered around the disc.

#### Figure 5: Computational domain for freestream case

For wind tunnel computations the same dimensions as the OJF were used except for the outlet part. The cooling system is not simulated but the outlet is extended 20\(R\) behind of the disc. The domain is composed of 20290 structured cells. The grid is stretched towards the boundaries (see Figure 6).

#### Figure 6: Computational domain for OJF

The thrust coefficient computed by using Blade Element Momentum code (BEM) is employed to calculate the pressure jump across the disk. BEM code is run at 720 RPM for the same rotor used in the wind tunnel measurements.

The pressure jump across the disc corresponds to the thrust,

\[ T = A(p^+ - p^-) \quad (1) \]

\[ \Delta p = \frac{C_T U^2 \rho}{A} \frac{A}{A} \quad (2) \]

where \(U_{\infty} = 10.77 \text{ m/s}\).

By keeping $U_{\infty}$ constant, Thrust Coefficient ($C_T$) is changed and the new pressure jump is calculated for different loadings. Performed 2D actuator disc simulations are tabulated in Table 2. For unsteady cases, time step, \(\Delta t\), is chosen as...
0.02 times of period. Period is calculated by keeping Strouhal number = 0.2. The simulations are performed till the convergence is obtained when the residuals reaches $10^{-6}$. For unsteady case, simulation is performed till the transient solution is converged.

<table>
<thead>
<tr>
<th>Case</th>
<th>Freestream case</th>
<th>Wind tunnel case</th>
</tr>
</thead>
<tbody>
<tr>
<td>C_T</td>
<td>C_T = 0.814, Pres. Jump = 57.8 Pa</td>
<td>C_T = 0.814, Pres. Jump = 57.8 Pa</td>
</tr>
<tr>
<td>C_T</td>
<td>C_T = 0.814, Pres. Jump = 63.9 Pa</td>
<td>C_T = 0.814, Pres. Jump = 57.8 Pa</td>
</tr>
<tr>
<td>1</td>
<td>Case 1 ✔</td>
<td>Case 2 ✔</td>
</tr>
<tr>
<td>4</td>
<td>Case 4 ✔</td>
<td>Case 5 ✔</td>
</tr>
<tr>
<td>3</td>
<td>Case 3 ✔</td>
<td>Case 7 ✔</td>
</tr>
<tr>
<td>6</td>
<td>Case 6 ✔</td>
<td>Case 8 ✔</td>
</tr>
</tbody>
</table>

Table 2: 2D AD Simulation cases, constant pressure jump

4 Results

This section presents experimental and computational results for 2D freestream and wind tunnel cases. Axial velocity, vorticity magnitude and induction factor are compared for different cases as tabulated in Table 2.

4.1 Experimental Results

In this part tip vortex core path and level of vortex wandering deduced from Stereo PIV images are presented in Figures 7-9. As a result of the centrifugal force acting on the particles, the particles are pushed away from the center. Therefore, in the PIV images the vortex core is clearly visible as a circular black spot. A visual inspection of the PIV images showed a noticeable movement of this spot within the ensemble. This phenomenon, known as vortex wandering. Vortex wandering, the slow side-to-side movement of the wing-tip vortex core, has been found to be a universal feature of wind-tunnel-generated wing-tip vortex structures [3]. The vortex core for each image is computed in order to draw an average path which reveals the wake expansion and to assess the level of wandering. This is observed as the vortex is transported downwind with a high expansion angle.

Figure 7: Traces of the tip vortices on each measurement plane with velocity vectors colored by velocity.

Figure 8: Tip vortex path with measurement planes.

Figure 9: Tip vortex wandering as average deviation from mean center.

4.2 2D Computational Results

Computational axial velocity and vorticity contour lines are compared with the experimental results
(see Figures 10 & 11 for freestream and wind tunnel cases). It is seen that the flow characteristics around the disc is different for freestream and wind tunnel cases. When the steady flow field is analyzed for wind tunnel cases at different $C_T$ values, it is observed that the CFD results are coming closer to the experiments (Black dots in Figures 10 & 11). However, it is known that the flow in the wake is unsteady. Therefore, unsteady simulations are performed at $C_T = 0.814$ (which is the expected $C_T$ value for experiments). It is seen that after the transition phase of the flow (at $t = 3.7s$), it reaches a convergence at $t = 26s$. The expansion at $t = 26s$ is less than the experiments. This result may be concluded that the loading in the experiments is higher than the expected. However, unsteady higher loading simulations is needed to come up a conclusion.

The induction factor is calculated by using the axial velocity distribution on the disk as formulated in Eq. 3.

$$u_{disk} = (1 - a)U_\infty$$

The difference between freestream and wind tunnel conditions is easily noticed from Figure 12. While the wind tunnel and freestream cases seem very close to each other at the lower $C_T$ value, at the higher values of $C_T$, the wind tunnel cases result in higher induction.

5 Conclusion

Wind tunnel experiment and CFD are used to investigate the expansion of the wake for a specific rotor model. Experimental work has been performed at the TU Delft Open Jet Facility by using Stereo PIV to study the aerodynamics of horizontal axis wind turbine. 2D actuator disc and 3D rotor simulations have been performed by using FLUENT. This paper represents the
answer of two objective questions of the study. The flow field around the prescribed model is affected by the geometry of the OJF since the flow in freestream case and wind tunnel case are different. By comparing the results of experiments with the results of CFD, one may conclude that the CFD modeling is very close to represent the flow behavior observed in the experiments. However, further investigation on unsteady simulations and 3D rotor simulation may be needed to come up a final conclusion on the disturbance of the wake.

References


