Numerical Investigations on the Fluid Behavior in the Near Wake of an Experimental Wind Turbine Model in the Presence of the Nacelle

A. Bouhelal1†, A. Smaili1, O. Guerri2 and C. Masson3

1 Laboratory of Green and Mechanical Development (LGMD), École Nationale Polytechnique, B.P. 182, El-Harrach, Algiers, 16200, Algeria
2 Renewable Energy Development Center (CDER), B.P. 62, Route de l’Observatoire, Bouzaréah, Algiers, Algeria
3 Department of Mechanical Engineering, École de Technologie Supérieure, 1100 Notre-Dame Ouest, H3C1K3, Montréal, Québec, Canada

†Corresponding Author Email: abdelhamid.bouhelal@g.enp.edu.dz
(Received June 16, 2022; accepted August 16, 2022)

ABSTRACT

Accurate predictions of the near wake of horizontal-axis wind turbines are critical in estimating and optimizing the energy production of wind farms. Consequently, accurate aerodynamic models of an isolated wind turbine are required. In this paper, the steady-state flow around an experimental horizontal-axis wind turbine (known as the MEXICO model) is investigated using full-geometry computational fluid dynamics (CFD) simulations. The simulations are performed using Reynolds-Averaged Navier-Stokes (RANS) equations in combination with the transitional k-kl-ω turbulence model. The multiple reference frame (MRF) approach is used to allow the rotation of the blades. The impacts of the nacelle and blade rotation on the induction region and near wake are highlighted. Simulation cases under attached and detached flow conditions with and without the nacelle were compared to the detailed particle image velocimetry (PIV) measurements. The axial and radial flow behaviors at the induction region have been analyzed in detail. This study attempts to highlight the nacelle effects on the near wake flow and on numerical prediction accuracy under various conditions, as well as the possible reasons for these effects. According to simulation results, the rotation of blades dominates the near wake region, and including the nacelle geometry can improve both axial and radial flow prediction accuracy by up to 15% at high wind speeds. At low wind speeds, the nacelle effects can be ignored. The presence of the nacelle has also been shown to increase flow separation at the trailing edges of the blade airfoils, increasing both root and tip vorticities. Finally, small nacelle diameters are recommended to reduce flow separation on the blades and increase the average velocity downstream of the rotor, thereby optimizing wind farm output power.

Keywords: Wind turbine; Aerodynamics; Near wake; Nacelle-blade interaction; CFD; Mexico.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cf</td>
<td>skin friction coefficient</td>
</tr>
<tr>
<td>(\vec{V})</td>
<td>velocity vector</td>
</tr>
<tr>
<td>(\bar{p})</td>
<td>average pressure</td>
</tr>
<tr>
<td>(\bar{u})</td>
<td>average axial velocity</td>
</tr>
<tr>
<td>(u_\infty)</td>
<td>freestream wind speed</td>
</tr>
<tr>
<td>(u_\tau)</td>
<td>friction velocity</td>
</tr>
<tr>
<td>(\bar{w})</td>
<td>average tangential velocity</td>
</tr>
<tr>
<td>(y^+)</td>
<td>dimensionless wall distance</td>
</tr>
<tr>
<td>(y_p)</td>
<td>first mesh element near the wall</td>
</tr>
<tr>
<td>(D)</td>
<td>rotor diameter</td>
</tr>
<tr>
<td>(F_N)</td>
<td>normal force</td>
</tr>
<tr>
<td>(k)</td>
<td>turbulence kinetic energy</td>
</tr>
<tr>
<td>(k_l)</td>
<td>laminar kinetic energy</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>TSR</td>
<td>Tip Speed Ratio</td>
</tr>
<tr>
<td>(\bar{v})</td>
<td>average radial velocity</td>
</tr>
<tr>
<td>(\tau_\infty)</td>
<td>wall shear stress</td>
</tr>
<tr>
<td>(\mu)</td>
<td>dynamic viscosity</td>
</tr>
<tr>
<td>(\rho)</td>
<td>fluid density</td>
</tr>
<tr>
<td>(\Gamma)</td>
<td>diffusion coefficient</td>
</tr>
<tr>
<td>(\phi)</td>
<td>general transport variable</td>
</tr>
</tbody>
</table>
1. INTRODUCTION

Due to the increasing demand for energy, especially clean energy, wind turbines are usually installed in large arrays or grids known as wind farms. In a wind farm, wind turbines can be placed in the wake of upstream turbines, producing 40% to 60% less power than in isolated conditions (Santoni et al. 2017). Therefore, accurately predicting the wake of an isolated wind turbine leads to good estimation and optimization of the wind farm's overall power production.

The wake developing downstream of a wind turbine can be divided into two regions: the near wake and the far wake. The far wake is located far from the wind turbine and immediately impacts the downstream turbines. The near wake is an induction region located downstream of a wind turbine at a distance of 1 to 5 times the rotor diameter (Abraham et al. 1999).

Accurate predictions of the near wake are critical in many onshore and offshore wind turbine applications, including but not limited to the following:

(i) Develop and improve fast engineering wake models (Bastankhah and Porté-Agel 2014; Madsen and Rasmussen 2004);
(ii) Increase the accuracy of the nacelle anemometry technology (Dahlberg et al. 1999; Smaili and Masson 2004), which is used as a practical way to estimate the wind farm output power;
(iii) Evaluate the available annual energy production in a given site (Feng et al. 2018);
(iv) Control and optimization of wind farms (Bartl and Sattran 2016);
(v) Optimize the aerodynamics-hydrodynamics interaction for offshore wind turbines (Gao et al. 2022);
(vi) Evaluate the maintenance testing and safety of wind turbine systems (Thomsen et al. 2007).

Different numerical methods are used to predict the near wake flow downstream of wind turbines, including the free/fixed vortex wake models (Sanderson et al. 2011; Tescione et al. 2016), actuator disc/surface/line models (De Cillis et al. 2021; Zhu et al. 2021; Masson and Smaili 2006; Masson et al. 2001; Sørensen et al. 1998) and the full-geometry CFD (computational fluid dynamics) model (Bouhelal et al. 2018; Sørensen et al. 2002). The full-geometry CFD model is thought to be more accurate than other methods because it does not require rotor data, unlike actuator disc/surface/line and vortex models (Hansen et al. 2006).

The geometry of the wind turbine, such as blades, nacelle, and tower, is responsible for the development of the near wake. To differing degrees, all these components (i.e., blades, nacelle, and tower), with their interactions with the fluid, contribute to the formation of the wake. Based on full-geometry CFD simulations, Zahle and Sørensen (2008) investigated the influence of the tower on the wake of a multi-megawatt wind turbine model. The nacelle and tower are found to create important quantities of turbulence in the near wake, considerably increasing the turbulent kinetic energy. The tower effect has been shown to have the least influence on the development of the wake. In comparison to the effect of the nacelle, the effect of the tower can be ignored. The same conclusion has been also demonstrated for small-scale wind turbines (Guo et al. 2021; Santoni et al. 2017).

Numerous studies have been carried out in order to better understand the role of the nacelle in the development of the near wake downstream of wind turbines. Micallet et al. (2013) demonstrated that radial flow dominates near wake flows and its complicated behavior is affected only by the blade geometry (Akay et al. 2014).

Many studies have shown that the nacelle effect is caused by flow separation in the rotor's root region and the formation of root vortices. Weihing et al. (2018) showed that increasing the diameter of the nacelle reduces the performance of the rotor at the root and increases flow separation. Abraham et al. (2019) observed, using an experimental study, that the turbulent kinetic energy field enhances turbulence in locations of high shear behind the blade tips and nacelle, and decreases turbulence behind the tower. Based on large eddy simulations, Zhu et al. (2022) concluded that the nacelle and tower increase the instability of the tip vortices.

Guo et al. (2021) showed that the increase in vorticities and turbulent kinetic energy produced by a wind turbine in the presence of a nacelle is the primary cause of rotor output power fluctuations. These fluctuations lead to an increase in fatigue loads and thus reduce wind turbine life operation.

It has been proven from the literature that both steady (Bouhelal et al. 2018) and unsteady (Regodeseves and Morros 2021) full-geometry CFD simulations produce good predictions of aerodynamic performance and velocities in the near wake. For both cases, the Reynolds-Averaged Navier-Stokes (RANS) approach is widely used for turbulence modeling. Sørensen et al. (2016) investigated the transition effects in RANS turbulence models, and it has been demonstrated that the transition effects can improve the accuracy of the predictions.

Reliable experimental measurements are required to validate numerical wake models. The measurements carried out in Europe's largest wind tunnel under a project known as MEXICO (Model rotor EXperiments In COntrolled conditions) (Boorsma and Schepers 2014; Snel et al. 2007) are probably the most widely used among researchers. The MEXICO project provides measurements of aerodynamics and velocities in the wake under various conditions.

From the literature, the nacelle modeling leads to an improvement in the wake prediction accuracy. However, the magnitude of this improvement and its relation to the tip speed ratio (TSR) are not well
understood. This research advances our understanding of the physics of near wake flows downstream of wind turbines. It also aims to investigate the fluid flow features in the induction region by quantifying the effect of blade rotations in the presence of the nacelle. For this purpose, a full-geometry CFD model has been proposed to simulate the steady-state flow around the MEXICO wind turbine using a transitional turbulence model (namely the k-kl-ω turbulence model).

In order to investigate the effect of the nacelle on the near wake downstream of the MEXICO wind turbine, simulated flow at attached and detached conditions with and without the nacelle was compared to detailed PIV measurements for different TSR (varying from 4 to 10).

The following questions are addressed in this study: (i) to what extent can the presence of the nacelle affect the evolution of the fluid behind the rotor and thus the accuracy of the near wake predictions? (ii) how does this effect relate to TSR? The investigation also looks into the possible causes of the nacelle effect by analyzing the axial and radial behavior of the flow, as well as the flow separation in the blade under various conditions.

The following sections highlight the MEXICO wind turbine model, PIV measurements, numerical method, and simulation results.

## 2. Experimental Description

In this study, the MEXICO wind turbine model was used since it provides experimental data for wind speeds in the near wake. The data is provided through the MexNext project's collaboration (Boorsma and Schepers 2014). The measurements were carried out in the German Dutch wind tunnels in 2014.

### 2.1 Wind turbine Model Description

MEXICO is a three-bladed, upwind, pitch control wind turbine. The diameter of the rotor is 4.5 m, while the height of the tower is 5.12 m. MEXICO model is characterized by a huge nacelle, measuring 4.4 m in length and 0.54 m in diameter. The blade geometry was created using three different airfoils (see Fig. 1). The blades are twisted and have a non-uniform chord (see Fig. 2).

### 2.2 Test Cases Description

Three axial upstream wind speeds have been tested, including low speed (TSR = 10), design speed (TSR = 6.7), and flow separation speed (TSR = 4). Particle Image Velocimetry (PIV) stereos were used to measure the axial and radial velocities components in three directions (x, y, and z) in the near wake at the 9 o’clock horizontal plane.

Velocity profiles were measured in different axial and radial transverse positions as shown in Fig. 3. The axial lines cover a distance of 10 m from upstream to downstream, whereas the radial lines cover about 3 m in the near wake. In addition, about 150 dynamic pressure sensors have been used to measure the rotor loads. For more detail, see (Boorsma and Schepers 2014; Schepers et al. 2014).

### 3. Numerical Method

#### 3.1 Computational Domain

Two computational domain configurations have been created with and without the nacelle using the ANSYS DesignModeler tool to investigate the effect of the nacelle on the near wake. The first configuration includes the entire rotor geometry (i.e., blades, hub, and nacelle), whereas the second is a reduced configuration that only includes an isolated blade. The second configuration thus represents one-

Fig. 4. Computational domain and mesh for the two studied configurations: Rotor (with the nacelle) and Blade (without the nacelle).

third of the full configuration. In the rest of this paper, these two configurations will be referred to as "Rotor" and "Blade" configurations respectively.

The computational domain, for both configurations, represents a cylinder 7.5xD in length and 5xD in diameter, where D is the rotor diameter. Cylindrical configurations were created around the blade and rotor in order to separate the rotating domain from the stationary domain and to increase the mesh elements in the near wake (see Fig. A).

3.2 Mesh Sensitivity Test

The solid-fluid interaction region is the most critical area that must be treated with attention during mesh generation (i.e., the boundary layer region). In this study, the height of the first mesh element ($y_p$) near the wall is estimated using the inverse $y^+$ relationship given by:

$$y_p = \frac{y^+ \mu}{u_t \rho}$$  \hspace{1cm} (1)

Where $u_t$ is the friction velocity defined by $u_t = \sqrt{\tau_w/\rho}$, and $\tau_w$ is the wall shear stress linked to the skin friction coefficient ($C_f$) as follows:

$$\tau_w = \frac{1}{2} C_f \rho U_*^2$$  \hspace{1cm} (2)

Here, $y^*$ is set to 1.0 in Eq. 1, and $C_f$ has been calculated using the Schlichting correlation (Schlichting et al. 1960) given by:

$$C_f = [2 \log_{10}(Re) - 0.65]^{-2.3}$$  \hspace{1cm} (3)

where $Re$ is the Reynolds number based on the radius of the blade R (i.e., $Re = \rho U_* R/\mu$).

Based on the first element height ($y_p$), 15 structured mesh layers have been created near the blade/nacelle walls for spatial resolution of the boundary layer. A mesh sensitivity test has been carried out for the Blade configuration. The Rotor configuration was then given the same mesh parameters. Three mesh cases have been considered, representing coarse (2.3 million nodes), medium (5.1 million nodes) and fine mesh (12 million nodes).

Figure 5 shows the comparison of the normal force distribution simulated at the design condition (i.e., TSR = 6.7). As indicated in Fig. 5, the results of the medium and fine meshes are very close to each other. While the results of the coarse mesh were far away, especially in the blade tip region. In addition, to properly examine the mesh effect, the mechanical

The power for the three mesh cases has been computed and presented in the Table 1.

As it can be seen in Table 1, the absolute error difference between the medium and fine meshes is less than 1%. Accordingly, the medium mesh has been used for the Blade configuration. For the Rotor configuration, the same mesh features were used.

For the two configurations (Blade and Rotor configurations), the average size of the elements in the rotating and stationary parts was 0.02 and 0.05 m, respectively. The Blade configuration mesh has around 5.1 million nodes, whereas the Rotor configuration mesh has 38 nodes.

### Table 1: Mechanical power for the three meshes

<table>
<thead>
<tr>
<th>Mesh case</th>
<th>Node number (×10⁶)</th>
<th>Power (kW)</th>
<th>Absolute Error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>2.314206</td>
<td>12.121</td>
<td>14.10</td>
</tr>
<tr>
<td>Medium</td>
<td>5.191501</td>
<td>15.066</td>
<td>06.76</td>
</tr>
<tr>
<td>Fine</td>
<td>12.305786</td>
<td>14.912</td>
<td>05.67</td>
</tr>
</tbody>
</table>

### 3.3 Mathematical Model

Steady-state numerical simulations of the flow around the MEXICO wind turbine were performed in this work for two configurations (Blade and Rotor configurations) using an incompressible finite volume CFD solver; ANSYS Fluent. The k-kl-ω (Walters and Cokljat 2008) transitional turbulence model was used to solve the RANS equations. This model is a low Reynolds model based on three transport equations, namely turbulent kinetic energy (k), laminar kinetic energy (kl), and the inverse turbulent time scale. The governing equations can be expressed using a general transport equation for a general variable \( \phi \) as follows:

\[
\bar{\nabla} \cdot \left( \rho \bar{\nabla} \phi \right) = \bar{\nabla} \cdot \left( \Gamma \bar{\nabla} \phi \right) + S_\phi
\]

In Eq. (4), the left term represents convection, the first right term represents diffusion, \( \Gamma \) is the diffusion coefficient, and \( S_\phi \) is a source term. Table 2 summarizes the governing equations to be solved in the CFD solver.

In Table 2, \( G \) and \( Y \) are respectively the production and the dissipation of energy (for \( k \), \( k_l \) and \( \omega \)), \( \alpha_T \) is the turbulent scalar diffusivity. For further information about the k-kl-ω model, see (Walters and Cokljat 2008).

### Table 2: Governing equations

<table>
<thead>
<tr>
<th>Equation</th>
<th>( \phi )</th>
<th>( \Gamma )</th>
<th>( S_\phi )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Continuity</td>
<td>1</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>x- momentum</td>
<td>( \bar{u} )</td>
<td>( \mu )</td>
<td>( -\frac{\partial \bar{p}}{\partial x} )</td>
</tr>
<tr>
<td>y- momentum</td>
<td>( \bar{v} )</td>
<td>( \mu )</td>
<td>( -\frac{\partial \bar{p}}{\partial y} )</td>
</tr>
<tr>
<td>z- momentum</td>
<td>( \bar{w} )</td>
<td>( \mu )</td>
<td>( -\frac{\partial \bar{p}}{\partial z} - \rho g )</td>
</tr>
<tr>
<td>Turbulent kinetic energy</td>
<td>( k )</td>
<td>( \nu )</td>
<td>( G_k - Y_k )</td>
</tr>
<tr>
<td>Laminar kinetic energy</td>
<td>( k_l )</td>
<td>( \mu + \rho \alpha_T )</td>
<td>( C_{kl} - Y_{kl} )</td>
</tr>
<tr>
<td>Inverse turbulent time scale</td>
<td>( \omega )</td>
<td>( \mu )</td>
<td>( \alpha_T )</td>
</tr>
</tbody>
</table>

\( \frac{\alpha_T}{1.17} \)

### 3.4 Boundary Conditions

The boundary conditions that have been used in this work can be summarized as follows:

(i) uniform axial velocity and low turbulence intensity at the inlet boundary.

(ii) Atmospheric pressure at the outlet boundary.

(iii) An interface technique was applied to separate rotating parts from stationary parts.

(iv) In the case of the Blade configuration, the remaining blades were calculated using periodic boundary conditions, taking advantage of the MEXICO rotor geometry’s 120° symmetry.

(v) The no-slip condition implies that the flow velocity is zero on the blade/nacelle walls.

### 3.5 Numerical Simulation and Computational Time

In this study, the convective terms for all equations were discretized using the second order Upwind scheme and the diffusion terms are discretized using central differencing scheme.

All the steady-state simulations presented in this paper are computed by a parallel PC with 64 GB RAM and 6 x 3.31 GHz CPU.
To resolve the Pressure-Velocity coupling equations, the SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) (Patankar 2018) and Coupled (ANSYS Fluent 2017) algorithms were compared for the Blade configuration, and it was found that the Coupled algorithm can accelerate the convergence of solutions and is more stable. The solutions converge with the SIMPLE algorithm for about 4500 iterations, and the computation time takes 3 days. On the other hand, when the Coupled algorithm is used, the solutions converge after 200 iterations, with a computation time of 22 hours. Therefore, the Coupled algorithm is recommended for this type of simulations (i.e., steady-state simulations of the flow around wind turbines). Noting that the Coupled algorithm resolves implicitly the continuity and the momentum equations.

4. RESULTS AND DISCUSSION

The flow around the MEXICO rotor is numerically simulated with three cases at axial wind velocities of 10 m/s (TSR = 10), 15 m/s (TSR = 6.7), and 24 m/s (TSR = 4) for the two studied configurations (without and with nacelle). The three averaged components of axial (\(\bar{u}\)), tangential (\(\bar{\theta}\)), and radial (\(\bar{r}\)) velocities in the near wake were computed and compared with PIV measurements to assess the influence of nacelle on the near wake.

As indicated in Fig. 3, the axial and radial velocities were extracted at the 9 o’clock position in the horizontal plane. The axial velocities were measured in two traverses’ positions: inboard part (\(y = 0.5\) m) and outboard part (\(y = 1.5\) m) of the blade, and the radial velocities were measured upstream and downstream of the rotor, at the positions \(x = +0.3\) m and \(x = -0.3\) m respectively.

4.1 Axial Flow Behaviors

Let’s begin with the axial flow behavior in the near wake. Figure 7 shows the distribution of the normalized velocities along the inboard (\(y = 0.5\) m) and outboard (\(y = 1.5\) m) traverses, computed and measured at the design condition (TSR = 6.7). The differences in maximum and minimum absolute errors for the axial velocity component (\(\bar{u}\)) between the Blade and Rotor configurations were 6.2059\% and 0.1889\%, respectively. This means that introducing the nacelle geometry into account in CFD simulations can improve axial velocity prediction by around 6\%.

The same fluid behavior was also noticed for the radial and tangential velocity profiles. Oscillations occur for all velocity components due to vortices generated by the blades’ rotation, the radial and tangential velocities began to gradually appear to have a maximum variation at the level of the rotor, and this variation disappeared progressively until it re-stabilized at zero in the downstream.

The simulation results were in good agreement with the PIV measurements for the two studied configurations (with and without nacelle). The difference in maximum and minimum absolute errors for the axial velocity component (\(\bar{u}\)) between the Blade and Rotor configurations was 6.2059\% and 0.1889\%, respectively. This means that taking nacelle geometry into account in CFD simulations can improve axial velocity prediction by around 6\%.

The difference in maximum and minimum absolute errors for the axial velocity component (\(\bar{u}\)) between the Blade and Rotor configurations, in this case, was 6.2059\% and 0.1889\%, respectively. The results showed that introducing the nacelle geometry into...
CFD improves near wake prediction by about 5%. Since the inboard traverses are not available in MEXICO tests. Figure 8 shows only the velocity components normalized by the free wind speeds along the outboard line (y = 1.5 m) for the two off-design conditions (TSR = 4 and 10). For the lowest wind speed (TSR = 10), the two configurations agree well with the experimental data. The difference in absolute error between the two configurations does not exceed 1.1%. In the other hand, for the highest wind speed (TSR = 4), where the flow is separated, the maximum absolute error difference between the two configurations is greater than 15%. The presence of boundary layer separation on the nacelle and blade walls complicates the flow. The Rotor configuration results are in good agreement with the experimental data and closely follow the velocity oscillations downstream of the rotor.

The axial velocity contours for different wind speeds have been plotted in Fig. 9 to properly assess the fluid structure and behavior of the axial flow in the near wake. It can be seen that the flow's interaction...
with the rotor reduces the velocity of the fluid downstream (i.e., the induction region), and this region decreases with the augmentation of wind speed for both configurations.

As shown in Fig. 9, the fluid is subjected to excessive acceleration in the root area of the blades in the case of the turbine without the nacelle. Previous research has observed this phenomenon (Guo et al. 2021; Zheng et al. 2018).

It can also be seen from Fig. 9 that, in addition to blade tip vortices, the presence of the nacelle creates root vortices near the nacelle wall and corrects the velocity induction behind the rotor, especially for design and high wind speeds. At low wind speeds, the prediction of wake flow fields was nearly identical for both configurations.

4.2 Radial Flow Behaviors

Following the comparison of axial flow, the emphasis is now on radial flow behavior. Figs. 10, 11, and 12 show the normalized velocity components at the upstream (x = -0.3 m) and downstream (x = +0.3 m) traverses near the rotor for both studied configurations (with and without the nacelle) for the three conditions TSR = 10, 6.7, and 4, respectively.

For the low wind speed (TSR = 10), as indicated in Fig. 10, the axial velocity decreases from the root to the tip of the blades due to the rotation of the blades for both upstream and downstream lines. After the fluid leaves the induction region, it returns to its initial velocity (free stream wind speed). Both configurations gave a good prediction of the width of the induction region. The greatest decrease in axial velocity for both numerical simulation and
experimental measurements was detected at $y/D = 0.45$ and $y/D = 0.5$ for upstream and downstream lines, respectively. In addition, due to the conservation of mass, the flow is forced to change its direction at the rotor level, which leads to an increase in its radial velocity ($\bar{v}$) and a decrease in its tangential velocity ($\bar{w}$). Both configurations (with and without the nacelle) produced accurate radial velocity predictions. However, the tangential velocity is slightly underestimated, particularly at the blade tips. Indeed, this is due to the well-known weakness of RANS/URANS turbulence models as reported in previous studies (Micallef 2012; Sørensen et al. 2014; Thé and Yu 2017).

With the increase of wind speed, as indicated in Figs. 11 and 12, the point of greatest velocity decrease turns back toward the center of the rotor and its normalized magnitude decreases. The difference in prediction between the two configurations is more pronounced, especially at high wind speeds.

It can be seen from Figs. 10, 11 and 12 that, in the absence of the nacelle, the radial flow behavior is underpredicted at the tip and overpredicted at the root of the rotor. Therefore, the presence of the nacelle affects both tip and root flows.

The tip flow behavior refers to the separation resulting from the nacelle as indicated in the previous study done by (Akay et al. 2014). While the overprediction in the outer part of the blade is still unknown, it can be explained here that the presence of the nacelle causes large separation zones to reach the blade tip. This phenomenon will be demonstrated in the next section.

4.3 Flow Separation

As previously demonstrated by the results shown in the preceding sections, the presence of the nacelle improves the near wake prediction, especially at high wind speeds, while the nacelle effect can be neglected at low wind speeds. The purpose of this section is to figure out why the nacelle improves the near wake prediction and to reveal the physics behind the nacelle effect on the near wake under various attached and detached flow conditions.

Figs. 13 and 14 show the streamlines on the suction side of the blade (on the upper side downwind), as well as the streamline contours in the planes normal to the blade surface for the two studied configurations (with and without the nacelle), at five sections representing blade span locations of 0.25, 0.35, 0.60, 0.82, and 0.92. It can be observed from Figs. 13 and 14 that, at low and medium wind speeds ($\text{TSR} = 10$ and 6.7), the streamlines are parallel to each other, and the flow remains attached to the whole of the suction side of the blade as well as to the upper and lower sections of the airfoils. However, due to vortex shedding in the blade’s root, a slight radial component appears in this region for both configurations (with and without the nacelle). The results show that the angle of attack (angle between relative velocity direction and chord line) increases from tip to root of the blade in each case due to twist angle augmentation. Additionally, as the wind speed increases, this angle increases, and as a result, the separation appears at high wind speeds on the upper sections of airfoils (extrados) for both configurations (with and without the nacelle). It can be observed from Figs. 13 and 14 that, at low and medium wind speeds ($\text{TSR} = 10$ and 6.7), the streamlines are parallel to each other, and the flow remains attached to the whole of the suction side of the blade as well as to the upper and lower sections of the airfoils. However, due to vortex shedding in the blade’s root, a slight radial component appears in this region for both configurations (with and without the nacelle). The results show that the angle of attack (angle between relative velocity direction and chord line) increases from tip to root of the blade in each case due to twist angle augmentation. Additionally, as the wind speed increases, this angle increases, and as a result, the separation appears at high wind speeds on the upper sections of airfoils (extrados) for both configurations (with and without the nacelle).
Fig. 13. Streamlines at the upper (downwind) side of the blade and streamline contours at five span-wise sections simulated for the Rotor configuration case (with the nacelle) for three TSR.

Fig. 14. Streamlines at the upper (downwind) side of the blade and streamline contours at five span-wise sections simulated for the Blade configuration case (without the nacelle) for three TSR.
separation on the blades, resulting in stronger vorticity and, as a result, reduced velocity in the near wake at high wind speeds. At low wind speeds, however, there is no significant difference in near wake predictions between the Blade and Rotor configurations because the flow is mostly attached and there are no vorticities. This explains the near wake overprediction previously observed in Fig.9 for the isolated blade configuration at high wind speeds.

5. Conclusion

The purpose of this work is to deeply study the physics of near wake flows downstream of wind turbines as well as to evaluate the near wake predictions with and without the nacelle. To achieve this objective, the full-geometry CFD approach has been used to simulate the steady-state flow around an experimental wind turbine, namely the MEXICO model. The transitional three-equation k-kl-o turbulence model has been used to enclose the RANS equations. To assess the effect of the nacelle, two computational domains have been considered: (i) Rotor configuration including the whole rotor with the hub and nacelle; and (ii) Blade configuration representing only an isolated blade.

Three axial wind speeds, representing attached, design, and detached conditions, have been simulated. The velocity components in axial and radial lines are compared with the MEXICO measurements. Good agreements have been obtained for both configurations with and without the nacelle geometry. According to the obtained results, at low wind speed values, the nacelle geometry may enhance near wake numerical predictions by about 5%. At high wind speed values, however, the flow becomes more complicated, and the inclusion of the nacelle improves near wake predictions by increasing tip and root vortices near the wall. It has been shown that the rotor-nacelle interactions may improve the near wake predictions by up to 15% at high wind speeds, and the CFD results closely follow the velocity oscillations downstream of the rotor.

In addition to the effects of the nacelle that were concluded in previous studies, such as that the presence of the nacelle increases turbulence and turbulent kinetic energy and creates vortices in the root, new insights were addressed in this study, the most prominent of which are:

- At high tip speed ratios (or low wind speeds) where the flow is attached, the effect of the nacelle can be ignored.
- The presence of the nacelle enhances flow separation across the entire blade. At high wind speed values, the separation can reach the blade tip and intensify the tip vortices.
- The flow separation in the blade is less in the absence of the nacelle than in the presence of the nacelle, resulting in an overprediction of the velocity field behind the rotor. Therefore, it is recommended to use small nacelle diameters in order to reduce the flow separation at the blades and thus increase the velocity in the near wake and the output power of wind farms.

Finally, the main challenge remains evaluating how far the results of this work can be generalized by studying numerically and experimentally other wind turbine models of various scales in the future. In addition, future research should include unsteady simulations and POD (proper orthogonal decomposition) analyses to gain a better understanding of the near wake dynamics.

Acknowledgements

The data used in this paper were supplied by the consortium which carried out the EU FP5 project called MEXICO: “Model rotor Experiments In COntrolled conditions,” to which nine European partners contributed.

The support from Directorate-General for Scientific Research and Technological Development (DG-SRDT) of Algerian government is gratefully acknowledged.

We would like to express our heartfelt condolences to Pr. Christian Masson, the fourth author of this paper, who passed away recently.

References


Sørensen, J., W. Shen and X. Munduate (1998). Analysis of wake states by a full-field


