Wind turbine rotor simulation using the actuator disk and actuator line methods

M Tzimas and J Prospathopoulos

Laboratory of Aerodynamics, Department of Mechanical Engineering, National Technical University of Athens, 9 Heroon Polytechniou Ave., 15780 Zografou, Athens, Greece

E-mail: jprosp@fluid.mech.ntua.gr

Abstract. The present paper focuses on wind turbine rotor modeling for loads and wake flow prediction. Two steady-state models based on the actuator disk approach are considered, using either a uniform thrust or a blade element momentum calculation of the wind turbine loads. A third model is based on the unsteady-state actuator line approach. Predictions are compared with measurements in wind tunnel experiments and in atmospheric environment and the capabilities and weaknesses of the different models are addressed.

1. Introduction

Wind turbine wakes have attracted a lot of attention by the research community since they are characterized by momentum deficits and increased turbulence levels, which result in reduced power outputs of a wind farm and increased loading. The issue is magnified by the need for installing the machines as closely as possible to each other, trying to maximize the exploitation of the land in wind energy installations.

Over the last 25 years several models of varying complexity have been used to predict the flow field inside a wind turbine wake, ranging from the simple engineering up to the advanced CFD models. Engineering models are based on the analytical solution of the simplified momentum or mass conservation equations and need calibration with experimental data. They predict the centerline velocity deficit and the wake width and assume self-similar profiles in the far wake [1,2]. Boundary layer methodologies [3], parabolic approximation [4] and simplified forms of the axisymmetric Navier-Stokes equations in the far wake [5] were intermediate steps, until the development of computer technology made possible the numerical solution of the full three-dimensional Reynolds-averaged Navier-Stokes equations (3D RANS). However, even in the field of the 3D RANS, the complexity of the methodologies varies according to the wind turbine and turbulence modeling.

The simplest approach of wind turbine simulation is the actuator disc method referred to as the Froude’s [6] momentum theory. Depending on the available data, wind turbine loading may be either approximated as a constant thrust or calculated using the blade element theory evolved by Glauert [7]. In order to consider the unsteady character of the flow, the so-called actuator line technique can be implemented [8], in which loading is distributed along lines representing the blade forces. Actuator line forces are calculated at each time step using a blade-element approach and tabulated airfoil data. This method has been successfully used for the calculation of the flow field in wind farms [9]. In cases of three-dimensional dynamic phenomena such as flow separation, semi-empirical corrections for the lift and the drag on the blade sections have been proposed [10,11]. The most accurate approach is a
full representation of the wind turbine blades geometry, however, the simulation of the boundary layer development along the blade requires a fine mesh which increases the computational cost substantially.

Regarding turbulence modeling, most of the existing models are based on the Boussinesq hypothesis which introduces the turbulent viscosity concept. The latter is estimated either explicitly through an algebraic equation (zero-order models) or by solving one or two transport equations, such as the commonly used k-ε and k-ω models [12], where k is the kinetic energy of turbulence and ε or ω are the rate or specific rate of dissipation of k respectively. More complex turbulence models take into account the anisotropy of the tensor of Reynolds stresses solving one explicit or implicit equation [13, 14]. Finally, the most advanced, but also most expensive in terms of computational cost, turbulence models for unsteady flows use the Large Eddy Simulation technique, which takes into account the large scales of eddies [15].

In the present work, a compressible RANS solver along with the k-ω turbulence model is used for the prediction of the flow field in the wind turbine wake. Simulation of the wind turbine is made using the actuator disk and actuator line techniques. Section 2 describes the numerical solver and the different wind turbine simulation approaches. In section 3, the predictions of the different methods are presented and compared with measurements and predictions of other methodologies. Finally, the conclusions regarding the capabilities and weaknesses of the two techniques are summarized in section 4.

2. Numerical modeling

Numerical simulations were performed using MaPFlow, an in-house multiblock unsteady RANS solver for compressible flows, developed at NTUA [16]. MaPFlow applies the finite volume method on unstructured grids using a second order cell-centered discretization scheme and the Roe approximate Riemann solver for the convective fluxes. In time, a second order implicit scheme is implemented using dual stepping for facilitating convergence. In order to extent its use to low Mach flows, a preconditioning scheme has been added. There are two options regarding turbulence modeling: the k-ω SST model as formulated by Menter [17] and the Spalart Allmaras one equation model [12]. MaPflow can handle moving or deformable grids and has been parallelized using the Message Passing Interface (MPI).

In the present work, fully turbulent conditions for incompressible fluids were considered using a low Mach number of 0.1 with the proper pre-conditioning. The low Mach number was used to ensure near incompressible conditions on the flow. Constant angular velocity and blade pitch was considered for each case study. Turbulence was modeled using the k-ω SST model, where k, ω are the turbulent kinetic energy and its specific rate of dissipation respectively. For wind tunnel conditions, the standard values of the model constants were used:

\[
\sigma_{k1} = 0.85, \quad \sigma_{\omega 1} = 0.5, \quad \beta_k = 0.075, \quad \beta_\omega = 0.0828
\]
\[
\sigma_{k2} = 1.0, \quad \sigma_{\omega 2} = 0.856, \quad \beta' = 0.09, \quad \alpha_1 = 0.31
\] (1)

whereas for atmospheric conditions the \(\beta'\), \(\beta_k\), \(\beta_\omega\),\(\sigma_{k1}\) model constants have been modified to be in accordance with the friction velocity measurements in neutral stratification:

\[
\sigma_{k1} = 0.5, \quad \beta_1 = 0.0275, \quad \beta_2 = 0.03036, \quad \beta' = 0.033
\] (2)
2.1. Boundary conditions

2.1.1. Atmospheric simulations

For atmospheric simulations, neutral stratification was considered. At the inflow plane of the computational domain, the wind velocity, \( u \) and the \( k, \omega \) were prescribed by the known relationships of similarity theory:

\[
\begin{align*}
  u(z) &= \frac{u_*}{\kappa} \ln \left( \frac{z}{z_0} \right), \\
  k &= \frac{u_*^2}{\sqrt{\beta} z}, \\
  \omega &= \frac{|u_*|}{\sqrt{\beta} \kappa z}, \\
  z &< z_{bl}
\end{align*}
\]

\[
\begin{align*}
  u(z) &= u_*, \\
  k &= 0, \\
  \omega &= \frac{|u_*|}{\sqrt{\beta} \kappa z}, \\
  z &\geq z_{bl}
\end{align*}
\]

where \( u_* \) is the friction velocity, \( \kappa \) is the von-Karman constant, \( z \) is the height above ground, \( z_0 \) is the roughness length, \( z_{bl} \) is the boundary layer depth and \( u_\infty \) is the free stream velocity. The roughness length \( z_0 \) is calculated by the ambient turbulent intensity of the air, \( TI_{in} \), obtained from previous experiments:

\[
z_0 = z_{hub} \exp \left( \frac{-0.9895}{TI_{in}} \right)
\]

Neumann conditions of fully developed flow (zero first derivative) were imposed for the velocity components and the \( k, \omega \) on the side and outflow boundaries of the computational domain. On the top boundary, Dirichlet conditions were imposed to describe the undisturbed flow outside the atmospheric boundary layer:

\[
u = u_\infty, \quad v = w = 0, \quad k = 0, \quad \omega = 0\]

Velocity was set to zero on the ground due to the non-slip condition. Wall functions were employed to preserve the profiles of (3) on the first grid point above ground. Atmospheric pressure was set to the inflow, outflow, top and side boundaries.

2.1.2 Wind tunnel simulations

For wind tunnel simulations, a uniform inflow velocity profile \( u = u_\infty \) was considered. The \( k, \omega \) profiles were estimated by the turbulence intensity, \( TI_{in} \), and the length scale of turbulence, which is taken equal to the rotor diameter, \( D \):

\[
k = 1.5 \cdot TI_{in}^2 \cdot u_\infty^2, \quad \omega = k^{0.5} / D
\]

Neumann conditions of fully developed flow were imposed on the outflow boundary for the velocity components and the \( k, \omega \). Pressure is set equal to the ambient atmospheric level on both inflow and outflow boundaries. The non-slip condition yields zero velocity on the tunnel walls (terrain, side and top boundaries), while the following wall function is employed for the calculation of the velocity on the first grid point:

\[
\frac{u}{u_*} = \begin{cases} 
  y^+, & y^+ < 11.63 \\
  \frac{1}{\kappa} \ln y^+ + b, & y^+ \geq 11.63
\end{cases}
\]
where $b=5.5$ is the common value for wind tunnel flows. The dimensionless height $y^* = y \cdot u_c / \nu$ was estimated using an iterative Newton-Raphson method. The eddy viscosity, $\nu'$, and the $k$, $\omega$ values are calculated from the following relationships:

$$
\nu' = \kappa \cdot y^* \cdot \nu, \quad k = u_c^2 / \sqrt{\beta}, \quad \omega = \nu' k
$$

(8)

2.2. Computational domain and mesh

The dimensions of the computational domain are selected so that the boundary conditions, as defined in Section 2.1, are well imposed, and there is no blockage effect. In atmospheric simulations, the inflow boundary is positioned 10 diameters upstream of the wind turbine to ensure that the boundary layer is well shaped when the flow reaches the rotor. The outflow boundary is positioned 20 diameters downstream of the rotor so that the wake deficit has been completely neglected and the flow is fully developed. For the same reason, side boundaries are positioned at a distance of 10 diameters from both sides of the rotor. The height of the top boundary is taken greater than the boundary layer depth, to allow for the imposition of Dirichlet free stream conditions. In wind tunnel simulations, the inflow boundary is positioned 3 diameters upstream of the wind turbine, so that the flow remains uniform when reaches the rotor. The outflow boundary is positioned 20 diameters downstream of the turbine rotor to allow for the imposition of Neumann boundary conditions for the velocity (fully developed flow).

The numerical mesh consists of quadrilateral cells and is refined close to the walls and the wind turbine rotor. A grid independency study showed that a number of 51 grid points along the rotor diameter is sufficient. In atmospheric simulations, the distance of the first node above ground is $10 \, z_0$, while in wind tunnel simulations, the distance of the first node from the walls (top, bottom and side boundaries) is 0.01m. In the regions among the walls and the turbine rotor, subsequent coarsening and refinement of the grid lines is made using geometrical progressions of varying ratio. In the main flow direction, which is aligned with x-axis, a grid refinement is made from the inflow boundary to the rotor position, and then, the grid is coarsened from the rotor to the outflow boundary.

2.3. Wind turbine modeling

2.3.1. Actuator Disk with uniform thrust (AD-uniform CT)

The simplest approach to simulate the presence of a wind turbine is the actuator disk with uniform thrust. The rotor is modeled as a momentum absorber upon which a uniform distribution of axial force is applied. This force is prescribed from the actuator disk theory and is calculated using the thrust coefficient for a corresponding reference wind speed. The wind turbine rotor is discretized into a number of control volumes which act as momentum sinks through the actuator force:

$$
F_x = T = \frac{1}{2} \rho U_{ref}^2 C_T \Delta S
$$

(9)

where $T$ is the thrust exerted by the rotor, $\rho$ is the density of the air at 1.23 kg/m$^3$, $U_{ref}$ is the reference wind speed, $C_T$ is the thrust coefficient and $\Delta S$ is the surface of the control volume. In the case of a stand-alone wind turbine, $U_{ref}$ is the undisturbed wind speed at hub height. The work of the actuator force is used as an energy sink of the conservation of energy equation. A number of 20000 time steps is used for full convergence.

2.3.2. Actuator Disk using Blade Element Momentum Theory (AD-BEM)

When the blade geometry and the lift-drag coefficients, $C_{L}, C_{D}$, are known, the loads on the actuator disk can be estimated using the blade element momentum (BEM) theory. Calculation of the axial and circumferential loads distribution is possible through the tabulated data of the blade airfoils for
different angles of attack. According to the BEM theory the forces are distributed to annular rings, so a polar-type discretization is applied to the rotor disk. An interpolation to the Cartesian grid is then needed so that the proper loads contribution is transferred to the actuator disk control-volumes. The number of time steps to ensure full convergence of the problem is 20000. No root or tip corrections are applied in the approach.

The elementary lift and drag forces on the blade section are given by:

\[ dF_L = \frac{1}{2} \rho C_L U_{rel}^2 c \, dr \]  
\[ dF_D = \frac{1}{2} \rho C_D U_{rel}^2 c \, dr \]  

where \( U_{rel} \) is the local induced velocity calculated from the solver at the present time step, \( c \) is the chord of the blade section and \( d \) is the width of the annular ring. The axial and circumferential forces are calculated through the angle \( \phi \) between the induced flow and the plane of rotation:

\[ dF_x = dF_L \cos \phi + dF_D \sin \phi \]  
\[ dF_y = dF_L \sin \phi - dF_D \cos \phi \]  

where \( dF_x \) is the force in the \( x \) direction of the Cartesian grid and \( dF_y \) is the force in the plane of rotation. The circumferential force \( dF_y \) can be further analyzed to the \( y \) and \( z \) directions of the Cartesian grid:

\[ dF_y = dF_y \cos \psi \]  
\[ dF_z = dF_y \sin \psi \]  

where \( \psi \) is the azimuth angle of the blade.

If the rotor is composed by \( B \) blades, the forces per unit surface will be

\[ p_{x,s} = \frac{B \, dF_x}{2 \pi rd} \]  
\[ p_{y,s} = \frac{B \, dF_y}{2 \pi rd} \]  
\[ p_{z,s} = \frac{B \, dF_z}{2 \pi rd} \]  

where \( 2 \pi rd \) is the surface of the annular ring. For each quadrilateral control volume with dimensions \( \Delta x, \Delta y, \Delta z \), multiplication by the corresponding surface provides the actuator forces that will be added directly as sink terms to the right hand side of the momentum equations, while the work of the forces is added on the right hand side of the energy equation:

\[ F_{x,s} = \frac{B \, dF_x}{2 \pi rd} \Delta y \Delta z \]  
\[ F_{y,s} = \frac{B \, dF_y}{2 \pi rd} \Delta y \Delta z \]  
\[ F_{z,s} = \frac{B \, dF_z}{2 \pi rd} \Delta y \Delta z \]  

2.3.3. Actuator Line (AL)

The actuator line approach models the blades of a wind turbine as lines moving in time with a physical time step that corresponds to the azimuthal angle step. Provided that the number of time steps per period, \( nsteps \), is specified, the physical time step, \( dt \), is calculated as:

\[ dt = \frac{2 \pi}{\omega \cdot nsteps} \]  

where \( \omega \) is the angular velocity of the blade. In the present work, \( nsteps \) is taken equal to 360, in order to achieve a fine azimuthal angle step of 1°.

Each line is discretized into elements representing segments of the blades. As in the blade element approach, the forces on each segment are calculated via the lift and drag coefficients and are interpolated to the Cartesian grid to obtain the contributions to the control volumes (Eqs. (10)-(15)). These contributions are used directly as sink terms in right hand side of the momentum equations. Actuator line is an inherently unsteady approach. No root or tip corrections were applied. The work of
each force is added as sink term on the right hand side of the energy equation. As an unsteady approach more time steps are needed for convergence, thus reaching upwards of 150000 on any case study.

3. Application and results

Application of the method is made on three cases for which experimental data are available. The first one refers to measurements in real atmospheric conditions (Sexbierum wind farm), while the other two refer to wind tunnel experiments (MEXICO and NREL wind turbine models).

3.1. The Sexbierum wind turbine case

Measurements were carried out in 1992 at the Dutch Experimental Wind Farm at Sexbierum, in the northern part of The Netherlands [18]. The wind farm contains 18 wind turbines of 340 kW rated power with a rotor diameter of 30 m and a hub height of 35 m. There were two measurement campaigns: one focusing on the single wake, and the other on the double wake situation. In the present investigation, the single wake case is simulated, with a free stream velocity of 8.5 m/s and ambient turbulence intensity of 13.5%. The roughness length \( z_0 \) was estimated \( z_0 = 0.01765 \). The computational domain is discretized using a mesh of 1.3 million grid points, 130 along the main flow direction (x-axis) and 90-100 along the y and z directions.

In Figure 1 the velocity deficit predictions of MaPFlow using all three wind turbine models are compared against the experimental data, and the predictions of the CRES-flowNS solver. CRES-flowNS [19] is a 3D RANS solver for incompressible flows using the k-\( \omega \) turbulence model. Discretization is performed with the finite volume method using a coordinate transformation on a curvilinear mesh. The AD-uniform \( C_r \) method is applied in order to simulate the wind turbine rotor. In the near wake (2.5 diameters downstream) the deficit is significantly underestimated by all models due to the fact that the eddy-viscosity concept of the two-equation turbulence models generates a too high turbulent kinetic energy in the area surrounding the turbine [20]. The actuator line approach slightly improves the velocity deficit prediction. Although the rate of diffusion seems to be well predicted, the initial difference is retained further downstream. In the far wake, differences among the different models become minimal. The difference between the predictions of the AL with BEM and the AD models can be also observed on the velocity contours (Figs. 2, 3). At 2.5 diameters downstream of the rotor, differences in the pattern of the velocity contours can be seen, while at 5.5 diameters any difference has been neglected.

3.2. The MEXICO wind turbine model case

In the context of the MEXICO EU project [21] detailed aerodynamic measurements were carried out on a wind turbine model with a diameter of 4.5 m, which was placed in the largest European wind tunnel, the German Dutch Wind Tunnel, DNW with a size of 9.5 x 9.5 m\(^2\). Measurements were carried out at two rotational speeds, 325 and 425 rpm. In the present work, the case of 425 rpm is investigated for three wind velocities, 10, 14.7 and 24.12 m/s. The computational domain is discretized using 90 grid points along the y and z directions and 100 along the x direction, resulting in a mesh of 900000 grid points.

In Figure 4, the predicted axial force distributions using the AD-BEM and the AL turbine models are compared against the experimental data and the predictions of a CFD and a potential simulation which take into account the full blade geometry. The CFD simulation has been performed using MaPFlow (MaPFlow-blade3D), whereas the potential simulation has been performed using the GenUVP free wake solver [22]. GenUVP is an unsteady potential flow solver in which the effect of solid boundaries is represented by means of surface source and/or dipole distributions, while the
Figure 1. Near wake velocity deficit of Sexbierum wind turbine. 2.5D (left), 5.5D (center) and 8D downstream (right). The horizontal axis symbolizes the angle between wind direction and wind turbine-observer direction.

Figure 2. Velocity contours of Sexbierum wind turbine at 2.5D downstream. MaPFlow simulation using AD-BEM (left) and AL (right).

Figure 3. Velocity contours of Sexbierum wind turbine at 5.5D downstream. MaPFlow simulation using AD-BEM (left) and AL (right).
wakes are modelled by means of freely moving vortex blobs. Viscous corrections are taken into account through tabulated $C_L, C_D$ data for the various blade sections.

GENUVP and MaPFlow-blade 3D predictions, as well as the experimental data have been obtained through integration of the pressure distributions over the blade sections. However, due to the much fewer pressure ports in the measured pressure distribution and the use of a linear interpolation between the pressure ports this is a source of uncertainty of the integrated forces as discussed in [23]. CFD predictions of spanwise loads reduce when the experimental measurements resolution is used instead the available CFD resolution. On the other hand, the AL and AD-BEM results are obtained directly as the calculated forces on the blade.

At the low wind speed of 10m/s the AL approach well predicts the axial loading of the blades, whilst the AD-BEM approach overestimates the loading. GenUVP and MaPFlow-blade3D simulations also overestimate the loading. As wind speed increases, 3D phenomena, such as flow separation, are more pronounced, leading to increasing differences between the predictions of the AL, AD models and the experimental data. For the highest wind speed of 24.12 m/s these deviations become too large. On the other hand, the predictions of the MaPFlow-blade3D simulation are in good agreement with the measurements. GenUVP predictions also deviate significantly from the experimental data indicating that, at high velocities, a post-run viscous correction on loads using tabulated $C_L, C_D$ data is not sufficient. The differences between the predictions of the AL model and the experimental data can be attributed to the utilization of pre-calculated $C_L, C_D$ data for the loads calculation. These data have been derived using 2D simulations that do not account for the 3D flow effects on the blade, which have a significant contribution at high wind speeds. Furthermore, tip vortex and root corrections have not been taken into account.

![Figure 4. Distribution of forces on Mexico blade at 10m/s (a), 14.7 m/s (b) and 24.12 m/s (c).](image)
In Figure 5, the total thrust on the rotor, obtained by integrating the sectional forces along the blade, is presented. For the experimental data, only 5 radial stations have been used, considering a linear interpolation in between them, while for the predictions, the resolution of the simulations is much finer. This will cause an underestimation of the experimental loads estimated as 6% [23]. In Figure 5a, the experimental data have been corrected using a factor of 1.06 to be comparable to the predicted forces. Another approach, which is perhaps more accurate, is to use the same 5 radial positions for the integration of the predicted sectional loads as shown in Figure 5b.

Using either approach, the comparison of the axial forces presented in Figure 5 is consistent with the comparison of the sectional loads presented in Figure 4. At 10m/s wind speed, the total thrust is well predicted by the AL method, while the other two methods overestimate it. This is a result of the fact that the AL prediction of the sectional load is in better agreement with the measurements at the outer part of the blade (from 60% to 92%), as shown in Figure 4a. At the 14.7 m/s wind speed, the AL method underestimates the total thrust, while the MaPFlow-blade3D and the AD-BEM method overestimate it. Again, this difference is justified from the comparison of the sectional loads at the outer part of the blade: Figure 4b shows that the AL method underestimates the axial load at 60% and 82% span. The MaPFlow-blade3D code significantly overestimates the axial load at 60% span, whereas the AD-BEM method produces an almost linear variation with radius resulting in an increasing overestimation from 60% span and above. Finally, at 24.12 m/s wind speed, the agreement between the MaPFlow-blade3D prediction and the measured value is consistent with Figure 4c which shows that only a full 3D simulation is capable of reproducing the axial load variation. The good prediction of the integrated load by the AD-BEM method results from the fact that the underestimation of the loading at the inner part of the blade is compensated by the overestimation of the loading at the outer part.

**Figure 5.** Integrated axial forces at 10, 14.7 and 24.12 m/s. (a) Predictions have been integrated from the root to the tip using fine resolution. Experimental data is corrected by a factor of 1.06. (b) Predictions have been integrated using 5 radial positions to be comparable with experimental data.
For the methods simulating the full blade it is possible that there are differences from the real geometry at certain regions resulting in subsequent differences in the predictions. Such a region is the 60% span, at which a systematic deviation between the MaPFlow-blade3D predictions and the measurements is observed.

3.2. The NREL wind turbine model case

The Unsteady Aerodynamics Experiment (UAE), conducted by the National Renewable Energy Laboratory (NREL), was a test of an extensively instrumented wind turbine in the NASA-Ames 24.4m x 36.6m wind tunnel [24]. The wind turbine model was 2-bladed, with a diameter of 10.058m and a constant pitch of 3°. Measurements were carried out at a rotational speed of 72 rpm, for a wind velocity range between 7m/s and 25m/s. In the present work the cases of 7m/s and 15m/s are investigated. A mesh of 110x100x100 grid points is used for the discretization of the computational domain.

In Figure 6, the load coefficients predicted using the AL model are compared against the measurements and the predictions of the Ellipsys3D solver [25]. The Ellipsys3D solver models the full blade geometry using periodic boundary conditions to account for the second blade of the rotor. At the low wind speed of 7 m/s the normal to the chord force is well predicted along the main part of the blade, while differences shown close to the root and the tip are attributed to the 3D flow effects and the tip vortex respectively. The tangential to the chord force shows some larger deviations from the experimental data, but the agreement can be regarded as satisfactory. At the higher wind speed of 15m/s, the presence of strong flow separation results in larger deviations of the predicted $C_n$ close to the blade root, whilst the predicted $C_t$ differs significantly from measurements along the whole blade.

![Figure 6](image.jpg)

**Figure 6.** Normal forces coefficient (a),(c) and tangential forces coefficient (b),(d) on NREL blade at 7m/s (up) and 15 m/s (down).

4. Conclusions

The actuator disk and actuator line models were implemented into a compressible CFD solver to predict the loads and the velocity deficit in the wake of a stand-alone wind turbine rotor. In the
actuator disk approach forces were estimated using either a uniform thrust coefficient or the blade element momentum theory. Predictions were compared against experimental data and predictions of potential and CFD solvers which simulate the full blade geometry. At low wind speeds the rotor thrust was predicted satisfactorily, whereas the predictions of the tangential force presented larger deviations from the measurements. This is attributed to the fact that the estimation of the tangential force is more sensitive to the variations of the angle of attack, which is influenced from possible 3D phenomena. As the wind speed increased, such 3D effects and flow separation become more pronounced, resulting in larger deviations from the measurements, due to the fact that forces are calculated using 2D $C_L, C_D$ airfoil data. Predictions at the root and tip portions of a blade could be improved by introducing corrections for the effect of the flow separation at the root region and the tip vortex. In general, loading is better predicted using the actuator line instead of the actuator disk approach (the latter overestimates the loads), suggesting the importance of considering the unsteady character of the flow.

Regarding the predictions of the velocity deficit, a significant underestimation is observed in the near wake (2.5 diameters downstream of the rotor). This is a known weakness of the two-equation turbulence models which overestimate the normal stress and the production of turbulent kinetic energy. However, a small improvement was observed when implementing the actuator line instead of the actuator disk approach. It is possible that a denser mesh would result in a further improvement of the predictions. As a next step, a more advanced turbulence model, like the Large Eddy Simulation technique which is inherently unsteady, could be implemented into the CFD solver and combined with the actuator line wind turbine model.

References

[1] Lissaman, P.B.S., “Wind Turbine airfoils and rotor wakes,” in D. A. Spera (ed.), Wind Turbine Technology, ASME Press, New York, 1994, pp. 283-323
[2] Voutsinas, S.G., Rados, K.G. and Zervos, A., “On the Analysis of Wake Effects in Wind Parks,” Wind Engineering, 1990, 14, pp. 204-219
[3] Taylor, P. A., “On Wake Decay and Row Spacing for WECS Farms,” Proceedings of 3rd International Symposium on Wind Energy Systems, Lyngby, 1980, pp. 451-468
[4] Crespo, A., Manuel, F., Moreno, D., Fraga, E. and Hernández, J., “Numerical Analysis of Wind Turbine Wakes,” Proceedings of Delphi Workshop on Wind Energy Applications, Delphi, 1985, pp. 15-25
[5] Sforza, P. M., Stasi, W. and Smorto, M., “Three-dimensional Wakes of Simulated Wind Turbines,” AIAA Journal, 1981, 19, pp. 1101-1107
[6] Froude, R. E., “On the Part Played in Propulsion by Differences of Fluid Pressure”, Trans. Inst. Naval Architects, vol. 30, 1889.
[7] Glaüert H. Airplane Propellers, Vol IV, div. L. In: Durand WF, editor. Aerodynamic Theory. Berlin: Julius Springer, 1935
[8] Sørensen, J. N., and Shen, W. Z., “Numerical Modelling of Wind Turbines,” J. Fluids Engineering, 2002, 124, pp. 393-399
[9] Troldborg, N., “Actuator Line Modeling of Wind Turbine Wakes”, Ph.D. Dissertation, Dept. of Fluid Mechanics, Technical University of Denmark, DTU, 2008
[10] Snel, H., Houwink, R., J. Bosschers, W.J. Piers, “Sectional Prediction of 3-D Effects for separated flow on rotating blades”, Eighteenth European Rotorcraft Forum, Avignon, France, September 1992.
[11] Snel, H., Houwink, R., J. Bosschers, W.J. Piers, and A. Bruining, “Sectional Prediction of 3-D Effects for stalled flows on rotating blades and comparison with measurements”, Proc. of the ECWEC ’93 Conference, Travemunde, Germany, pp 395-399.

[12] Wilcox, D. C., Turbulence Modelling for CFD, DCW Industries Inc., La Canada, California, 1993, ISBN 0-9636051-0-0, 80-90.

[13] Gatski, T.B., Wallin, S., “Extending the weak-equilibrium condition for algebraic Reynolds stress models to rotating and curved flows.” J. Fluid Mech., 2004, 518, 147–155.

[14] Franke, M., Wallin, S., Thiele, F., “Assessment of explicit algebraic Reynolds-stress turbulence models in aerodynamic computations”. Aerospace Science and Technology, 2005, 9, 573–581.

[15] Bechmann, A. and Sorensen, N.N., “Hybrid RANS/LES method for wind flow over complex terrain.” Wind Energy, 2010, 13(1), 36-50.

[16] Papadakis, G., “Development of a hybrid compressible vortex particle method and application to external problems including helicopter flows”, Ph.D. Dissertation, Aerodynamics Laboratory, National Technical University of Athens, 2014, p. 7

[17] Menter, F.R., “Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications,” AIAA Journal, Vol. 32, No. 8, Aug. 1994, pp. 1598-1605.

[18] Cleijne, J.W., “Results of the Sexbierum Wind Farm; Single Wake Measurements”, TNO Report No.93-082 for JOUR-0087 project, 1993.

[19] Prospathopoulos, J., Politis, E. S., Rados, K. G, Chaviaropoulos, P. K., “Evaluation of the effects of turbulence model enhancements on wind turbine wake predictions”, Wind Energy, 14(2), pp.285-300, DOI: 10.1002/we.419, 2010.

[20] Réthoré, P.-E., “Wind Turbine Wake in Atmospheric Turbulence”, PhD thesis, Risø National Laboratory for Sustainable Energy, 2009

[21] Mexnext (Phase 1) “Analysis of Mexico wind tunnel measurements”, Final report of IEA Task 29.

[22] Voutsinas, S.G., “Vortex Methods in Aeronautics: How to make things work”, Int. Journal of Computational Fluid Dynamics, Vol 20, No 1, 2006.

[23] Madsen, H., Prospathopoulos, J., Voutsinas, S., Riziotis, V., Diakakis, K., Chassapoiannis, P., Gomez-Iradi, S., Echarri, X., Ruiz, A., Shen, W., Sørensen, N., “Validation of high rotational speed aerodynamics by wind tunnel tests”, Deliverable 2.13 Part-II of the INNWIND E.U. project, 7th FP, June 2015.

[24] Simms, D., Schreck, S., Hand, M., Fingersh, L.J., “NREL Unsteady Aerodynamics Experiment in the NASA-Ames Wind Tunnel: A Comparison of Predictions to Measurements”, Technical Report NREL/TP-500-29494, National Renewable Energy Laboratory, Colorado, USA, 2001.

[25] Sørensen, N. N., Michelsen, J., Schreck, A. S., “Navier–Stokes Predictions of the NREL Phase VI Rotor in the NASA Ames 80 ft d 120 ft Wind Tunnel”, Wind Energy. 2002.