Actuator Line Simulation of Wake of Wind Turbine Operating in Turbulent Inflow

To cite this article: Niels Troldborg et al 2007 J. Phys.: Conf. Ser. 75 012063

View the article online for updates and enhancements.
Actuator Line Simulation of Wake of Wind Turbine Operating in Turbulent Inflow

Niels Troldborg, Jens N. Sørensen, Robert Mikkelsen
Dep. of Mechanical Engineering, Fluid Mechanics Section, Build 403, Technical University of Denmark, 2800 Lyngby.

Email: nt@mek.dtu.dk

Abstract. The wake of a wind turbine operating in an atmospheric turbulent inflow without mean shear is simulated using a numerical method, which combines large eddy simulations with an actuator line technique. A turbulent inflow with the same spectral characteristics as the atmosphere is produced by introducing time varying body forces in a plane upstream the rotor. The results of the simulation are compared to those obtained on a wind turbine in uniform inflow at the same mean wind speed and from this comparison a number of features of the influence of inflow turbulence on wake dynamics are deduced. Furthermore, the results are used to verify the validity of some of the basic assumptions employed in simpler engineering models and to study their bounds of application. The large amount of data from the wake simulation can easily be used in simple engineering methods to model a wind turbine operating in the wake of an upstream turbine

1. Introduction

Wind turbines located in wind farms are subject to the unsteady wake flow of upstream turbines, which strongly affects fatigue loading and power production. These “shadow” effects are one of the main reasons that the fundamental features of wakes have been a topic of intensive research, both experimentally and numerically during the last decades as is revealed from the comprehensive review on wind turbine wakes by Vermeer et al. [1]. However, as noted in this review, the basis of wind turbine aerodynamics is still not fully understood even under simple operational conditions. For wind turbines operating in an unsteady environment due to atmospheric turbulence, wind shear, wind direction change, terrain effects and the wake of other wind turbines knowledge of the wake properties is even more limited.

The usual modelling approach used by industry for compensating for wake interaction is by increasing the overall turbulence level in the design process, however, with the disadvantages of not capturing the impact of the wake deficit from upstream turbines. Nor does such a modelling approach satisfactorily take into account the important mechanism of wake deficit meandering, which has been observed in field measurements. It is greatly acknowledged and also confirmed in measurements by Madsen et al. [2] that this dynamical large scale phenomenon may contribute significantly to the increased loads of turbines, which are operating in the wake of other turbines. For this reason there is a

1 To whom any correspondence should be addressed
widespread desire in the wind industry to get more reliable predictions of the flow field experienced by a wind turbine operating in the wake of other wind turbines.

Thomsen et al. [3] interpreted the meandering of wind turbine wakes as being attributed to the traverse velocity fluctuations of the ambient wind field and has developed simple models based on this approach. This interpretation was supported by Bingöl et al. [4] who recently presented LiDAR measurements of the dynamics of the wake from a full scale wind turbine located in the field. However, in a recent wind tunnel study of the wake flow characteristics behind a two bladed wind turbine, Medici et al. [5] observed a clear indication of large scale vortex shedding similar to the well known von Karman vortex street experienced behind two-dimensional cylinders and they hypothesized that these findings might be connected with wake meandering.

A common difficulty in field measurements is that the dynamically changing inflow generally is not fully known and even in situations where turbulence is low data acquisition is limited to rather few measuring stations. Therefore, it can be difficult to identify and isolate the effect of a specific process; though with the emergence of the LiDAR technique some of these problems can be greatly reduced.

On the other hand research work conducted in wind tunnels under controlled conditions often suffer from various scale and blocking effects whereby the findings of such studies might not relate to the “real worlds” full scale wind turbines. The NREL Unsteady Aerodynamics Experiment [6,7] and the recent European MEXICO project are, so far, some of the very few wind tunnel experiments carried on wind turbines which are representative to full scale turbines. However, in the former project only few wake measurements was conducted and the data from the MEXICO project still remains to be processed.

Numerical simulations of rotor wakes is a valuable supplement to the various measurements because such computations can provide all the information needed about the wake. Furthermore, in computational fluid dynamics (CFD) it is easy to study the effect of changing a single parameter. Most numerical studies on wind turbines only consider uniform inflow and assume steady state conditions. Furthermore, they often do not resolve the wake to a degree where the generated flow structures, like the tip vortices, are preserved sufficiently far downstream the turbine.

Recently, however Zahle [8] presented full three-dimensional unsteady Navier Stokes computations on the NREL wind turbine using the so-called overset grid method to facilitate a high resolution of both rotor and wake with a reasonable number of grid points. Good agreement with measurements was obtained including the blade tower vortex interaction.

Generalized actuator disc and related methods, which combines full set Navier Stokes equations with a blade element approach and two-dimensional airfoil data, is another promising method for studying the dynamics in the wake since the influence of the blades on the flow field is modelled with a low amount of mesh points. This type of methods has been used by several researchers for both stand alone turbines [9, 10 and 11] and rows of wind turbines [12]. Computations have mostly been carried out assuming steady state or uniform inflow but also a few studies have been conducted in simulated atmospheric boundary layers.

Alinot et al. [13] used the actuator disc technique and a modified k-ε turbulence model to simulate a wind turbine operating in an atmospheric boundary layer with different types of stratification; however, the computations were carried out using a fairly coarse resolution.

Mikkelsen et al. [14] presented actuator line and disc computations on both isolated turbines and rows of wind turbines placed in arbitrary prescribed velocity profiles using a simple method based on introducing body forces in the entire computational domain.

Whereas, some numerical work on wind turbine wakes in sheared inflow exists, to the authors knowledge no one has to date attempted to simulate the effect of atmospheric turbulence on wind turbine wake dynamics. Even though there is a lot of literature on generating turbulent inflow for LES and DNS related industrial problems [15] these techniques has so far not been used in studies on wind turbines.

Comprehensive numerical simulation of the atmospheric boundary layer has been undertaken by e.g. Bechmann [16] who recently developed a LES based model for simulating atmospheric turbulence
over complex terrain and used it with success in three atmospheric test cases. However, such works still have not been combined with studies on wind turbine wakes.

This paper presents numerical simulations of the unsteady wake of a wind turbine in atmospheric turbulence without wind shear using the actuator line technique. The paper is an extension of the work recently presented by Troldborg et al. [17] where actuator line computations were used for simulating the wake of a wind turbine in uniform inflow at different tip speed ratios. The turbulent inflow is modelled using a simple approach where synthetic atmospheric turbulent fluctuations are superimposed to the mean flow upstream the rotor.

The objective of the paper is to study the influence of inflow turbulence on wake dynamics and to verify the validity of the basic assumptions employed in various simpler models.

2. Methods

2.1. Actuator line method

The flow field around the wind turbine rotor was simulated using the actuator line model developed by Sørensen and Shen [18]. This model combines a three-dimensional Navier-Stokes solver with a technique in which body forces are distributed radially along lines representing the blades of the wind turbine. Thus, the flow field around and downstream the wind turbine is governed by full three-dimensional Navier-Stokes simulations, while the influence of the rotating blades on the flow field is computed by calculating the local angle of attack and then determining the local forces using tabulated airfoil data.

\[
f_\varepsilon = f \otimes \eta_\varepsilon, \quad \eta_\varepsilon(d) = \varepsilon^{-2} \pi^{-3/2} \exp \left[ -\left( \frac{d}{\varepsilon} \right)^2 \right]
\]  

(1)

Here \(d\) is the distance between cell centred grid points and points at the actuator line and \(\varepsilon\) is a parameter that serves to adjust the concentration of the regularized load.

The main advantage of representing the blades by its airfoil data is mentioned in the introduction that much fewer grid points are needed to capture the influence of the blades compared to what would be needed for simulating the actual geometry of the blades. Therefore, the actuator line model is well suited for wake studies since grid points can be concentrated in a larger part of the wake while keeping the computing costs at a reasonable level. On the other hand, the models reliance on tabulated 2D airfoil data makes it greatly dependent on both their quality as well as the method used for modelling the influence of dynamically changing angles of attack and stall. However, as the main purpose of the present paper is to study fundamental wake effects it is considered of minor importance to capture the loads on the rotor exactly.

The applied blade forces needs to be distributed smoothly on several mesh points in order to avoid singular behaviour. In practice the aerodynamic blade forces are distributed along and away from the actuator lines in a three-dimensional Gaussian manner by taking the convolution of the computed local load, \(f\), and a regularization kernel \(\eta_\varepsilon\) as shown below.

2.2. Flow solver

The computations of the global flow field were carried out using the 3D flow solver EllipSys3D developed by Michelsen [19, 20] and Sørensen [21]. This code solves the discretized incompressible Navier-Stokes equations in general curvilinear coordinates using a block structured finite volume approach. EllipSys3D is formulated in primitive variables (pressure-velocity) in a non-staggered grid arrangement. In EllipSys3D the solution to the Navier-Stokes equations is advanced in time using an iterative time-stepping method. In each time step a number of sub iterations are carried out where the momentum equations are used as a predictor and the rewritten continuity equation (pressure correction equation) is used as a corrector for the solution at the subsequent time step. The pressure correction equation was solved using the PISO algorithm and pressure decoupling is avoided using the Rhie/Chow interpolation technique. The convective terms were discretized using a hybrid scheme.
combining the third order accurate QUICK (10%) scheme and the fourth order CDS scheme (90%). This scheme was employed as a compromise between avoiding the unphysical numerical wiggles, occurring when using the fourth order CDS and limiting numerical diffusion due to the upwinding nature of the QUICK.

Large eddy simulation (LES) was used to model the small length scales of turbulence. In LES the governing equations are obtained by filtering the time dependent Navier-Stokes equations in physical space such that those eddies which are below a certain size are filtered out. The resulting equations thus only govern the dynamics of the large scales, while the smaller scales are modelled by some additional terms, the so-called subgrid terms. In the present paper the subgrid scale (SGS) viscosity was modelled using a mixed scale model, which is presented in Sagaut [22]. Here the SGS viscosity is given by

$$\nu_{SGS}(x,t) = \rho C_m \left[ \nabla \times \tilde{u}(x,t) \right] \left[ \left( \frac{q_i}{q_j} \right)^{\frac{1}{2}} (x,t) \right]$$

Here \( \rho \) is the density, \( \tilde{u} \) is the filtered velocity, \( \tilde{A} \) is the filter cut-off length, which is set equal to \( \Delta V_{Vol}^{1/3} \), where \( \Delta V_{Vol} \) is the volume of a given cell and \( C_m \) and \( \alpha \) are constants which are respectively 0.01 and 0.5. The kinetic energy is evaluated in physical space as

$$q_i^2(x,t) = \frac{1}{2} \left( \tilde{u}_i(x,t) \right)^2$$

The mixed scale model has been chosen since it benefits from both accounting for the dissipation of energy and the important interaction between the smallest resolved scales and the largest unresolved scales.

2.3. Modelling atmospheric turbulence

The influence of atmospheric turbulence on the rotor wake was simulated using a technique where synthetic turbulent fluctuations from a pre-generated turbulence field are introduced to the mean flow in a plane upstream the rotor. In essence this method corresponds to introducing a grid in front of the wind turbine and let the generated turbulent structures move downstream over the rotor by convection.

2.3.1. Generation of inflow turbulence

The input turbulence field was generated using the algorithm by Mann [23, 24], which is also currently used in the aero-elastic code HAWC. This algorithm is based on a model of the spectral tensor and is capable of simulating all three components of a three-dimensional incompressible turbulence field. Furthermore, it can simulate turbulence with the same second order statistics as the atmosphere. However as discussed by Bechmann [16] the modeled turbulence field lacks the two-point correlations and the derivative skewness equals zero.

The spectral tensor (three-dimensional spectrum) is modeled from the linearized Navier Stokes equation combined with an assumption of linear shear and a model for eddy lifetime. Thereby, the model is formulated in terms of the von Karman spectral tensor for isotropic turbulence and a parameter \( \Gamma \) describing the anisotropy of the flow. Hence, in total the model of the spectral tensor contains only four adjustable parameters, which besides \( \Gamma \), is a turbulent length scale, \( L_t \), the three dimensional Kolmogorov, \( \alpha \) and the dissipation of turbulent kinetic energy \( \varepsilon \). These parameters are subsequently determined such as to fit the spectral tensor to commonly used spectral models for a given mean wind speed \( V_{\infty} \), height above ground and roughness length \( y_0 \). The roughness length is defined from the logarithmic mean wind profile:

$$V_z(y) = \frac{u_f}{\kappa} \left[ \ln \left( \frac{y}{y_0} \right) + 34.5 \frac{f y}{u_f} \right]$$

Here \( \kappa = 0.4 \) is the von Karman constant and \( u_f \) is the friction velocity.

The output of the algorithm is a box of equidistantly spaced turbulence. Here, the z-axis is in the direction of the mean wind speed and is inferred as a time axis via Taylor’s frozen turbulence hypotheses. It should be noted that since the simulated turbulent fluctuations are periodic in all directions turbulence was generated in a box with each of the cross flow dimensions twice the size.
2.3.2. Applying the Turbulent Fluctuations

The common approach when applying turbulent fluctuations from a pre-generated pseudo turbulence field is simply to superimpose the fluctuating velocities to the mean velocities at the inlet boundary. Here, however, it is proposed to use a method based on the immersed boundary technique whereby inflow turbulence is imposed by introducing time varying body forces in a plane upstream the rotor. Traditionally the immersed boundary technique is used to deal with boundaries in complex geometries since it offers a simple way of coping with boundaries that do not comply with the mesh layout. A review of the various applications of the immersed boundary technique is provided by Reck [25]. The reason for using this approach here is mainly to avoid possible problems with lack of continuity; Even though the turbulence generator by Mann automatically produces incompressible fields, continuity is generally not conserved in a discretized domain. This is not a problem when used in an aero-elastic model but can be problematic in a numerical simulation and therefore introducing the synthetic turbulence in terms of body forces rather than as a mass source/sink seems beneficial.

The idea behind the immersed boundary technique is simply to obtain a desired velocity in a given grid cell by applying an appropriate body force to the considered cell. Considering the incompressible Navier-Stokes equations

\[ \frac{\partial \vec{u}}{\partial t} + \nabla(\vec{u}\vec{u}) = -\frac{1}{\rho} \nabla \rho + \nu \nabla^2 \vec{u} + \vec{f}, \quad \nabla \cdot \vec{u} = 0 \]  

As mentioned above EllipSys3D solves this coupled set of equations using the SIMPLE/PISO predictor corrector method.

In discretized form the momentum equation yields a large system of linear equations

\[ A_p \vec{u}_{p}^{t+1} + \sum_i A_i \vec{u}_{i}^{t+1} = \vec{S}_p + \vec{f}_p \]  

Here \( \vec{u} \) denotes the velocity vector, \( S \) is a source term which includes the pressure and body forces and \( f \) refer to the external body force which is to be adjusted to establish a desired velocity. The subscript \( P \) denotes the current cell and the identifier, \( i = \{E, W, N, S, B, T\} \), the neighbouring cells.

From the discretized momentum equation the external force vector required to establish a fluctuation \( \vec{u}' \) about the mean \( \vec{u}_\infty \) in the computational cell \( P \) at a given time step is predicted as

\[ \vec{f}_p = A_p (\vec{u}_\infty + \vec{u}') + \sum_i A_i \vec{u}_i^{t+1} - \vec{S}_p \]  

During the following sub-iterations the instantaneous solution will converge towards the desired velocity.

The above procedure is carried out at each time step for all mesh points in a given plane upstream the rotor. To circumvent possible problems of singular behaviour the applied forces are smeared in the direction normal to the plane using a one-dimensional Gaussian approach. Hence the forces are distributed away from the plane by using the convolution

\[ f_z = f_p \otimes \eta_z; \quad \eta_z(d) = e^{-d^2/\varepsilon^2} \exp \left[ -\left( \frac{z-z_{plane}}{\varepsilon} \right)^2 \right] \]  

Again \( \varepsilon \) is a parameter that serves to adjust the concentration of the regularized load and \( z-z_{plane} \) is the normal distance from a grid point to the turbulence plane. In the present work the parameter \( \varepsilon = \Delta z \), the side length of a grid cell.

Since the resolution of the grid used for generating the turbulence was coarser than the grid used in the subsequent numerical simulation of the wind turbine both spatial and temporal interpolation is required. For a given time \( t \) in the simulation, two successive planes from the turbulence box corresponding to the turbulent field at time \( T_1 \) and \( T_{i+1} \), where \( T_1 < t < T_{i+1} \) is used as input. For each of
these planes bilinear interpolation is used to get the velocity in point P at respectively time $T_i$ and $T_{i+1}$. Thereafter these velocities are interpolated in time to get the velocity at time $t$.

2.4. Computational domain
The computations were conducted in a computational mesh identical to the one used in [17] and is sketched in figure 1. The domain is Cartesian with $L_x \times L_y \times L_z$ being $18R \times 18R \times 26.8R$, where $R$ is the rotor radius.

![Figure 1: Sketch of the computational domain and used coordinate system](image)

The actuator lines were located 7 rotor radii downstream the inlet with the point of rotation in the centre and the turbulence plane was located 1 rotor radii upstream the actuator lines. A high concentration of grid points were distributed equidistantly in the region around and downstream the rotor, see figure 1 in order to preserve the generated flow structures in the wake. Outside the equidistant region grid points were stretched away towards the outer boundaries. The grid was divided into 32 blocks (2 in the x and y direction respectively and 8 in the z-direction) with 64 grid points in each direction. Thus the resulting grid consisted of 128 grid points in the x and y direction respectively and 512 grid points in the z-direction. With the used grid configurations a rotor radii was resolved with 30 grid cells in the equidistant region. In the computations the following boundary conditions were applied:

- At the inlet boundary ($z/R=-9R$) the velocity in the z-direction was assumed uniform and equal to the free stream velocity, i.e. $V_z = V_\infty$, while the velocity components in the x and y direction were zero, i.e. $V_x = V_y = 0$.
- At the outlet ($z/R=17.8$) zero velocity gradient was imposed, i.e. $\frac{\partial V_x}{\partial z} = \frac{\partial V_y}{\partial z} = \frac{\partial V_z}{\partial z} = 0$.
- At the lateral boundaries ($x/R=-9, x/R=9$) periodicity were imposed.
- At the lower ($y/R=-9$) and upper ($y/R=9$) boundaries symmetry conditions were imposed i.e. $V_y = 0$ and $\frac{\partial V_x}{\partial y} = \frac{\partial V_z}{\partial y} = 0$.

2.5. Wind turbine and flow parameters
The computations were conducted using airfoil data from the Tjæreborg wind turbine. The blade radius of this turbine is 30.56 m and it rotates at 22.1 RPM, corresponding to a tip speed of 70.7 m/s. The blade sections consist of NACA 44xx airfoils with a chord length of 0.9 m at the tip, increasing
linearly to 3.3 m at hub radius 6 m. The blades are linearly twisted 1° per 3 m. The technical details of the rotor can be found in Øye [26]. All computations presented in this paper were carried out at a Reynolds number of $10^5$ based on rotor radius. This Reynolds number is considerably lower than experienced in the field and thus has a significant impact on the computed turbulent length scales in the wake. However, as discussed by Sørensen et al. [11] the Reynolds number is expected only to have a minor influence on the overall wake behavior provided that it has reached a certain critical minimum.

3. Validation of numerical method

The actuator line computations have already been thoroughly validated in [9, 10, and 17] and therefore only the method of producing turbulent inflow will be validated in this section.

In the validation study turbulence was produced in a plane located in a uniform flow field and the development of the field was studied in a number of downstream sections. Obviously, downstream the turbulence plane fluctuations will decay since no turbulence is produced to balance the dissipation and therefore the main objective of the present study is to quantify the rate of turbulence decay.

The turbulence field used for the investigation was generated using the algorithm by Mann [23, 24] assuming a mean wind speed of $V_\infty = 10$ m/s, a height above ground of $y = 60$ m and a roughness height of $y_0 = 0.05$. The given parameters correspond to a friction velocity and a $V_Z$ turbulence intensity of approximately respectively 0.54 and 0.14. The generated turbulence field was furthermore fitted to all three components of the Kaimal spectra $S$, which in its two-sided form is given by:

$$S_{Vx} f = \frac{8.5n}{(1 + 9.5n)^{5/3}}, \quad S_{Vy} f = \frac{1.05n}{1 + 5.3n^{5/3}}, \quad S_{Vz} f = \frac{52.5n}{(1 + 33n)^{5/3}} \quad (9)$$

Here $f$ is the frequency and $n = f y / V_\infty$.

The dimensions of the used turbulence box was $(L_x \times L_y \times L_z) 8R \times 8R \times 128R$ and from this domain a box of dimensions $4R \times 4R \times 128R$ was extracted to avoid problems related to the periodicity of the turbulence field. The number of grid points in the final turbulence box was $64 \times 64 \times 2048$ resulting in a completely grid with a resolution corresponding to 16 grid point per rotor radii. The turbulence field was introduced in a z-plane of size $4R \times 4R$ located 6 rotor radii downstream the inlet and with its center point in the centre of the domain.

Figure 2 compares the theoretical Kaimal spectra with respectively the spectra computed directly from the input turbulence field produced by the Mann algorithm (left) and the spectra obtained at three different sections downstream the grid (right). Note that the spectra are averages over the entire domain and is presented in non-dimensional form. Figure 2a shows the initial spectral characteristics of the turbulence field. As seen the resemblance between the theoretical and computed spectra is generally good over most of the frequency range. The reason that the computed turbulence field has somewhat lower energy at the highest frequencies is that the turbulence wind field is spatially averaged over each grid cell. Figure 2b shows the evolvement of the spectral characteristics with downstream position. As seen the spectra obtained 1 rotor radii downstream the turbulence plane is characterized by having a rather steep slope in the inertial sub range, however, further downstream the slope apparently return to the theoretical value. This transient behavior occurs because the input turbulence field is not a solution the full Navier Stokes equations and hence needs to adapt to the numerical solver.

From figure 2 it seems like the decay of turbulence is rather low but it is somewhat difficult to see. A clearer plot is obtained by integrating the one dimensional spectra over the entire frequency range to obtain an estimate of the variance of the turbulent velocity, i.e.

$$\text{var}(V) = 2 \int 0^\infty S_V df$$

Here the 2 is included because the spectra considered here are two-sided.
Figure 3 shows the estimated variance of each velocity component as a function of the distance to the turbulence plane. As seen the variance of the $V_Z$ component (in the flow direction) decreases continuously with downstream position and reaches a level of approximately 50% of the initial value. The variance of the $V_X$ component initially drops rapidly and then reaches a rather constant value, whereas the last component stays nearly constant over the entire region.

Figure 2: Comparison of the analytical Kaimal spectrum with the spectrum of the introduced turbulence field. a) Initial spectral characteristics. b) Development of one dimensional spectrum.
4. Results & discussion
In this section some of the results from the simulation of the wake generated behind the Tjæreborg wind turbine operating in an unsteady inflow are presented. In the computation the mean wind speed was $V_\infty = 10$ m/s corresponding to a tip speed ratio of 7.07. The wake of the turbine operating in uniform inflow at the same tip speed ratio has already been studied previously [17] and thus a comparison can reveal the overall influence of the unsteady inflow on the wake. As mentioned earlier the inflow turbulence field was introduced in a plane located 1 rotor radii upstream the turbine and was identical to the one used in the validation study in order to enable a direct comparison.

Figure 4 compares the development of the wake from the turbine operating in respectively uniform and turbulent inflow by displaying instantaneous contours of the absolute vorticity in a vertical slice intersecting the wind turbine centre axis.

![Figure 4: Development of mean variance of each velocity component](image)

Figure 3: Development of mean variance of each velocity component

![Figure 4: Downstream development of the wake visualized using vorticity contours. The rotor is located to the left. a) Uniform inflow. b) Turbulent Inflow.](image)
For the turbine operating in a uniform inflow the wake is dominated by a system of root and tip vortices that remain stable until approximately 10 rotor radii downstream, where an instability occurs and the wake subsequently breaks up and becomes unstable. Hence in this situation the first 10 rotor radii of the wake are well defined by the tip vortex sheet. When the turbine is exposed to an unsteady inflow the wake development is observed to undergo significant changes. In this case the external turbulent fluctuations perturb the vortex system, whereby the wake becomes unstable much closer to the rotor. From the visualization it is possible to identify various coherent flow structures in the wake, the largest of which are clearly much larger than the diameter of the rotor and appear as a meandering of the wake. Organized structures due to the presence of the tip and root vortices can be distinguished up to approximately 6 rotor radii behind the rotor but further downstream the interior of the wake seems to be fully turbulent. It is interesting to observe that at the shown instant there is a region approximately 7 rotor radii downstream where the wake apparently undergoes a contraction so that the radial extent of the wake at this point appears smaller than the rotor diameter. The same observation was made in the work of Binöl et al. [4] and is most likely due to a combination of large scale out of plane motion and stretching of the wake. The stretching of the wake is more apparent in figure 5, which shows vorticity contours at 3 different downstream cross sections in the wake.

Figure 5: Planar vorticity contours respectively 2 (a), 6 (b) and 10 (c) rotor radii behind the turbine

From figure 5 it is further seen that the tip and root vortices are unstable already 2 rotor radii downstream the turbine but at least the presence of the tip vortices can still be identified 6 rotor radii downstream. In the section 10 rotor radii downstream the wake has clearly broken down to conventional small scale turbulence.

In figure 6 the development of the axial velocity in the wake is shown and compared to the corresponding results in uniform inflow. The shown profiles are averaged in both time and in the circumferential direction.

As seen the wake of the rotor in uniform inflow is characterized by having a nearly constant velocity over most of the radial distance, which indicate a wake governed by the induction of stable tip and root vortices. On the other hand, the wake of the rotor operating in a turbulent inflow is seen to undergo a rapid transition into a bell shaped velocity deficit indicating that the wake becomes dominated by small scale turbulence. However, this wake shape could also be partly attributed to large scale wake meandering – indeed a sharper wake deficit might be found if the computed average velocity profiles were based on a point following the centre of the wake.

Figure 7 shows the azimuthally and temporally averaged normalized tangential velocity profiles at various downstream sections for both the tested cases. The figure clearly reveals that, while swirl in the wake is significant for all downstream section in the uniform inflow case, it decays rapidly towards zero when inflow turbulence is present. The quantity presented in figure 7 is also a measure of the amount of circulation in the wake. In [17] it was found that circulation is fairly conserved in the wake of a rotor in uniform inflow as long as viscous phenomena are not too dominant. However, the rapid
decay of the tangential velocity when turbulence is introduced to the inflow indicates that circulation is generally not conserved in the wake.

Figure 6: Downstream development of the axial velocity deficit for the turbine in respectively a uniform (full line) and a turbulent inflow (dashed line).

Figure 7: Averaged tangential velocity profiles at different downstream positions for the rotor operating respectively in uniform inflow (full line) and turbulent inflow (dashed line).

Figure 8 presents the downstream development of the rms. fluctuating axial velocity for the rotor operating respectively in uniform and turbulent inflow. In the wake of the rotor operating in uniform inflow turbulence is mainly present in the region of the root and tip vortices but as the wake develops turbulence increases and become distributed over a larger part of the wake. When inflow turbulence is included the axial rms. velocity is generally higher and initially also peaks near the region of the tip vortices. It is interesting to note that at 14 rotor radii downstream the rotor the maximum axial rms. velocity is actually larger when inflow turbulence is not included. From figure 8 it is revealed that assuming the total turbulence level in the wake to be equal to the sum of the turbulence induced by the wind turbine and the turbulence from the atmospheric flow does not necessarily yield good results.
Figure 8: Averaged axial rms. velocity profiles at different downstream positions for the rotor operating respectively in uniform inflow (full line) and turbulent inflow (dashed line).

Figure 9 shows the spectral characteristics at different downstream sections in the wake of the turbine subject to turbulent inflow. The shown spectra are averages of a number of points inside the wake. For comparison purposes the analytical spectra of Kaimal is also shown.

As seen the shape of the spectra is similar to the Kaimal spectra, though as expected the energy level in the wake is higher than outside the wake over most of the frequency range. The figure further
reveals that the turbulence in the wake becomes increasingly isotropic as it moves downstream. This is in good agreement with observations made in the field, see e.g. [1] for a review.

In order to make some initial investigations of large scale wake dynamics the average axial velocity of a number of points in a cross section inside the wake was computed and plotted as a function of time. The result is seen in figure 10 for two cross-sections located respectively 2 and 10 rotor radii downstream the wind turbine. Also included in the figure are the corresponding averages of the axial velocity when no turbine is present as well as the averaged input signal from the Mann algorithm.

![Figure 10: Temporal variation of the spatially averaged axial velocity inside the wake for cross-sections located respectively 2 rotor radii (a) and 10 rotor radii (b) downstream the wind turbine.](image)

The apparently strong correlation between the shown signals suggests that the large scale motion of the wake at least partly is governed by the large eddies in the inflow turbulence. This conclusion is in good agreement with the finding of Bingöl et al. [4]. The reason for the phase shift between the input signal and the other signals is the spatial displacement between the two considered cross-sections and the turbulence plane.

5. Conclusion
The wake of the Tjæreborg wind turbine operating in an atmospheric turbulent flow without shear has been simulated using a numerical method, where large eddy simulations are coupled with an actuator line technique. The method is further combined with a model where a turbulent inflow with the same second order statistics as the atmosphere is generated by introducing time varying body forces in a plane upstream the rotor.

Through a comparison with a similar simulation carried out on the same wind turbine subject to uniform inflow the influence of atmospheric turbulence on the wake dynamics has been studied. The study indicated that estimating the turbulence level in the wake of a turbine in unsteady inflow by adding wind turbine induced turbulence and atmospheric turbulence in a simple manner does not necessarily yield good results.

Furthermore, it has been shown that inside the far wake turbulence tends to be more isotropic than outside.

Finally, the investigation suggested that a main factor governing the large scale motion of the wake is the large coherent structures of the atmospheric turbulence.

From the simulation presented above a large database of snapshots obtained in different downstream cross-sections of the wake has been established. This data set easily casts itself into a form where it can be used in an aero-elastic simulation of a wind turbine operating in the wake of another turbine.
References
[1] Vermeer, L.J., Sørensen, J.N. and Crespo, A. "Wind Turbine Wake Aerodynamics". Progress in Aerospace Sciences, Vol. 39, pp. 467-510, 2003
[2] Madsen, H.A., Larsen, G.C. and Thomsen, K. “Wake flow characteristics in low ambient turbulence conditions”, Presented at Copenhagen Offshore Wind 2005, Risø National Laboratory, Denmark, 2005.
[3] Thomsen, K., Madsen, H.A. and Larsen, G.C. “A new method can predict detailed response for turbines in wind farms”, Fact sheet AED-RB-16(EN), Risø National Laboratory, Roskilde, Denmark, 2003.
[4] Bingöl, F. Mann, J. and Larsen, C.G. “Laser Measurements of Wake Dynamics” Scientific proceedings at EWEC, Milan, 2007.
[5] Medici, D., “Experimental Studies of Wind Turbine Wakes”, Doctoral Thesis in Fluid Mechanics, TRITA-MEK 2005:19, Stockholm, Sweden, 2005
[6] Fingersh, D., Simms, D. Hand, M., Jager, D., Cortrell, J., Robinson, M., Schreck, S. and Larwood, S. “Wind tunnel testing of NREL’s unsteady aerodynamics experiment” AIAA paper 2001-0335, 2001.
[7] Schreck, S. “The NREL full-scale wind tunnel experiment introduction to the special issue. Wind Energy, 5(2-3), 2002.
[8] Zahle, F. “Wind Turbine Aerodynamics Using an Incompressible Overset Grid Method. PhD thesis, Imperial College, London, 2006.
[9] Mikkelsen, R., “Actuator Disc Methods Applied to Wind Turbines”, MEK-FM-PHID 2003-02, 2003
[10] Ivanell, S.S.A., "Numerical Computations of Wind Turbine Wakes", Licentiate Thesis, TRITA-MEK, Technical Report 2005:10, Stockholm, Sweden, 2005
[11] Sørensen, J.N., Shen, W.Z. and Munduate, X. (1998) "Analysis of Wake States by a Full-field Actuator Disc Model". Wind Energy, Vol. 1, pp. 73-78, 1998
[12] Sørensen, J.N., Mikkelsen, R. and Trolodborg N. “Simulation and Modelling of Turbulence in Wind Farms” Scientific proceedings at EWEC, Milan, 2007.
[13] Alinot, C. and Masson, C. “Aerodynamic simulations of wind turbines operating in atmospheric boundary layer with various thermal stratifications” AIAA-2002-42, ASME Wind Energy Symposium, 40th collection of technical papers, Reno, January, 2002.
[14] Mikkelsen, R., Sørensen, J.N. and Trolodborg, N. "Prescribed Wind Shear Modelling Combined with the Actuator Line Technique". Presented at EWEC, Milan, 2007.
[15] Kempf, A., Klein, M. and Janicka, J. “Efficient Generation of Initial –and Inflow Conditions for Transient Turbulent Flows in Arbitrary Geometries” Flow, Turbulence and Combustion 74:67-84, Springer, 2005
[16] Bechmann, A. “Large-Eddy Simulation of Atmospheric Flow over Complex Terrain”, Risø-PhD-28(EN), 2007.
[17] Trolodborg, N., Sørensen, J.N. and Mikkelsen, R. "Numerical Simulations of Wake Characteristics of a Wind Turbine in Uniform Inflow” To be published in 2007.
[18] Sørensen, J.N. and Shen, W.Z. "Computation of Wind Turbine Wakes using Combined Navier-Stokes/Actuator-line Methodology". Proc. European Wind Energy Conference EWEC ’99, 56-159, Nice, 1999
[19] Michelsen, J.A., “Basis3D – a Platform for Development of Multiblock PDE Solvers”. Report AFM 92-05, Dept. of Fluid Mechanics, Technical University of Denmark, DTU, 1994.
[20] Michelsen, J.A., “Block Structured Multigrid Solution of 2D and 3D elliptic PDE’s”, Report AFM 94-06, Dept. of Fluid Mechanics, Technical University of Denmark, DTU, 1994.
[21] Sørensen, N.N., “General Purpose Flow Solver Applied to Flow over Hills”, PhD Dissertation, Risø-R-827(EN), Risø National Laboratory, Roskilde, Denmark, 1995.
[22] Saugaut, P., “Large Eddy Simulation for Incompressible Flows – An Introduction”, Springer, 2005
[23] Mann, J. “The spatial structure of neutral atmospheric surface-layer turbulence”. Journal of Fluid Mechanics, 273, 141-168, 1994.

[24] Mann, J. “Wind field simulation”. Prob. Engng. Mech. Vol. 13, No. 4, pp. 269-282, 1998.

[25] Reck, M. “Computational fluid dynamics, with detached eddy simulation and the immersed boundary technique, applied to oscillating airfoils and vortex generators”. PhD thesis, Department of Mechanical Engineering, Fluid Mechanics Section, 2004.

[26] Øye, S., “Wind Turbine: Dynamic Flow Measurement”, AFM Notat VK-233, Department of Fluid Mechanics, Technical University of Denmark, DTU, 1992