Comparison of wind turbine loads inside a wake between engineering model and CFD calculation

B Roscher¹, R Schelenz, A Werkmeister, Y Shkara, L Stössel, G Jacobs

RWTH Aachen University, Center for Wind Power Drives, Campus-Boulevard 61, 52074 Aachen, bjoern.roscher@cwd.rwth-aachen.de

¹corresponding author: bjoern.roscher@cwd.rwth-aachen.de

Abstract. Depending on location and inflow situations, a wind turbine experiences a variety of loads. Especially when considering that wind turbines are placed in a cluster. Inside wind farms, the wind turbines experience wake effects. This effect results in an increased turbulence intensity and therefore into higher loads onto the wind turbine components. Such increased load situation could lead to an early fatigue failure. For this reason, it is necessary to investigate the influence of wake effect on the wind turbine loading. Multiple wake models have been developed to represent and inspect the wake effects. This paper will utilize the method proposed by the IEC 61400 norm versus an inflow field generated by a Computational Fluid Model. The resulting 3D wind fields are used as an input into a multi-body simulation environment. This is done to determine the influence on the receiving wind turbine. The comparison will be inspected at the rotor hub coordinate system, as suggested by the Germanischen Lloyd, such that influencing parameter can be minimized.

It was determined that in case of a partial wake inflow the results of the CFD indicate a higher load variation in bending and tilt direction.

1. Introduction

Since 2017, it is mandatory to announce new wind farms via an auction system in Germany. Current bids indicate a high cost pressure (prices below 3,7 ct/kWh [1]), meaning that wind turbine must be used up to their full potential, especially while the lowest levelized cost of energy obtains the permission to construct. Additionally, potential wind energy areas are limited, such that turbines are placed rather close. Onshore the distance between wind turbines varies between 3 and 5 rotor diameters. Such a close spacing has certain disadvantages. While the wind turbine extracts energy out of the wind, the flow afterwards will be changed. The characteristics of such a downstream flow, also called wake effect, can be differentiated into two aspects. Firstly, the mean wind speed is reduced and secondly, the flow field will have an increased turbulence. These turbulences put wind turbine under a lot of stress, which is experiencing such a flow. To avoid any critical damage or early fatigue, the influence of wind turbines need to be known and evaluated.

A lot of efforts have been done, to develop models for the wake effects and their influences. In this paper two approaches will be used to determine a flow field that can be used for further analysis. The first approach will be based on the IEC 61400-1 norm [2]. The second approach utilizes a CFD output of a 3 MW wind turbine with a diameter of 126 m at a rated wind speed of 11 m/s. Both outputs will be used as an input into a multi-body simulation environment. The flow field will excite a generic CWD reference turbine equipped with a 126 m rotor [2]. The turbine will be located at a distance of 4 diameters. Finally, the load variation will be analyzed and quantified via a load distribution. To limit the
displayed work, the load will only be considered at the rotor hub. This component is directly associated with the aerodynamics forces and is connected to the drive train.

2. State of the Art of Wake Models

Wind turbine wake is typically classified into near wake and far wake, Figure 1. The near wake is expanded up to one diameter behind the turbine approximately. In this region the effect of the blade shape in the wake can be seen, leading to sharp velocity gradient. Tip and root vortices are created in the flow resulting in peak turbulent intensity. The turbine extracts the flow energy causing a jump in the pressure and velocity deficit downstream. The deficit of the velocity leads to shear layers separating the outer flow from the inside flow (behind the turbine). This layer thickens as it moves downstream. As the shear layer tends to meet the near wake characteristics, starting to dissipate and the far wake region begins. In the far wake region, the turbulence behaves like a mixer, the shape of the rotor can be felt as a reduction of axial velocity and an increase of turbulent intensity while the wake keeps expanding, [3, 4].

![Figure 1. Wind turbine wake [4].](image)

To study wind turbine wakes, both the wake and wind turbine models are important. Many engineering models have been used to estimate the wake aerodynamics. The simplest approaches are the engineering models like the Katic wake model that assume a self-similar wake propagation in the far wake downstream to predict the velocity deficit and turbulent intensity, [6-11].

The Blade Element Momentum (BEM) theory uses sectional blade aerodynamic characteristics (lift and drag coefficients) along with one dimensional momentum theory. It assumes a steady flow and an equilibrium of the momentum between the far up-and downstream and the forces on the blade. However, in practice the flow is unsteady. Furthermore, this method does not consider the Coriolis forces resulted from the blade rotation, [5].

Lifting line and vortex lattice methods assume an incompressible flow, like BEM method the lifting line method relay on the tabulated airfoils data to compute the vortex strength and the blade is presented as a line of bound vortices. In the more advanced method (vortex lattice) the surface of the blade is discretized to patches and the vortex elements are distributed along the chord instead of a simple line. Although this method does not take the drag into the account, it could predict the induced drag and the 3D effect is computed. The advantage of this method is it does not need the global momentum balance like the BEM, therefore it can solve for unsteady flow and the turbine yaw can be handled, these methods have been discussed in more details by [7, 8, 11, 12].

The previously mentioned methods serve for modelling wind turbine rotor and approximate prediction of the wake. To calculate the near and especially the far wake aerodynamics accurately, more advanced methods are needed.

The recently generalized actuator and direct methods are developed and commonly used, called computational fluid dynamics (CFD), to predict wind turbine wakes. Both methods solve Navier-Stokes equations to describe the flow. The flow domain is discretized to small grid elements, and the equations are solved for each element. The only difference between the two methods is the presentation of the
blade in the flow domain. While the blade geometry is physically modeled in the direct method, there is no real geometry for the blades in the actuator methods. In the later, the blade is replaced by its equivalent force and added to the momentum equation explicitly. Therefore, the needed grid size to represent the rotor is considerably lower which leads to less computational cost. The lack of this method lies in the need of the tabulated airfoil characteristics data, the lift and drag coefficients, which are hard to estimate (especially for 3D flow), [14, 15].

The simplest actuator model is the actuator disk. The actuator disk is seen as a semi-permeable surface where the average forces are projected on the flow forcing it to decelerate and a pressure jump appears downstream [4].

The actuator disk model is less accurate than the actuator line model, but it needs less computational resources. Recently Sørensen [14] introduced the actuator line approach, where the blade is represented as a line instead of disk. This model is widely used in the simulation of the wind farm combined with LES or DNS [14]. Some studies regarding this topic have been conducted by Shen et al. [17]. The concept of the actuator surface is more sophisticated than the actuator line. While the blade is presented as a line in the actuator line model, the actuator elements are distributed in two directions like a plate contains multiple points along the chord line, so that the flow cannot pass through the chord line. Knowledge of the skin friction and pressure distributions on the airfoil surface are necessary to use this model, [18, 19].

The direct method models the blade by constructing a physical grid fitted to the blade surface so that the flow is resolved near the surfaces of the boundary layer. The last method is the most accurate method to predict the wake, however the challenge using this model is to make a balance among the amount of the grid cells, the quality of the grid and the computational cost, this method has been frequently used in the simulation of wind turbine [20-23].

3. Simulation Approach
In this paper two methods of wake inflow simulation are used to determine their influence as an input into a Multi-Body Simulation. It is possible to determine the loads on a wind turbine located in wake inside a full CFD environment. However, CFD neglects deformations of a turbine and is time consuming. The purpose is to identify an inflow model that can be used in multiple load calculations. According to the IEC 61400-1 more than 1000 load calculations are required to certify a wind turbine [2]. Therefore, the load calculations are executed in a validated multi-body model (chapter 3.3). The CFD is used to achieve a high-fidelity representation of a wake situation. The authors are comparing the influence of different wake 3D wind field extracted out of a CFD simulation (chapter 3.1) versus a method proposed by the IEC (chapter 3.2). The setup is shown in Figure 2.

![Figure 2. Schematic representation of the simulation data flow](image)

3.1. CFD Model
The simulation is carried out using the software Ansys Fluent to solve the steady Navier-Stokes equations for the direct blade model. The multiple references to frame method is adopted, where the flow is rotating in the opposite direction of the rotor rotation instead of rotating the rotor itself. Using
this method there is no need to remesh for each time step. The blade model is the 5 MW NREL reference
turbine, it has a 61.5 m length and multi profile sections along the span and a spherical hub of 3 m
diameter [24]. The tower and the nacelle are not considered in the model.

The flow domain has a cylindrical shape and extended 2D upstream, 4D radial and 7.5D downstream
the rotor so that the boundary conditions do not affect the solution and allows accurate velocity
prediction in the location of the downstream turbine, Figure 3.

![Wind turbine model](Image)

![Refinement mesh domains](Image)

**Figure 3.** Computational domain

After creating the rotor geometry, it has been exported form the CAD environment to Fluent and
subtracted from the flow domain. The latter is then has been meshed using approximately \(5.4 \cdot 10^6\)
unstructured tetrahedral element. The mesh near the rotor surface is refined and additional prismatic
layers have been attached to capture the boundary layer, Figure 4.

![Computational mesh domain](Image)

**Figure 4.** Computational mesh domain

The k - \(\omega\) SST turbulent model is used to predict the wake, where k - \(\omega\) is solved near the walls and
k - \(\varepsilon\) away from the walls. The inlet is defined as a constant 11 m/s wind speed and a turbulent intensity
of 5 \%. Such a low turbulence intensity was considered such that the wake effect can be differentiated
from the ambient turbulence intensity. The outlet is set to ambient pressure, the surrounding to symmetry
and the rotor surface to the no-slip condition.

3.2. Wind field generation

The undisturbed inflow field, used as a reference, will be generated using TurbSim [25]. The tool is a
stochastic full-field turbulent wind generator. The turbulence in TurbSim is determined in the frequency
domain as a combination of velocity spectra and spatial coherence. Through an inverse Fourier transform a time series is obtained.

The underlying spectral model is the Kaimal Model, as proposed by IEC 61400-1. It assumes a neutral atmospheric stability. The spatial coherence is suggested by the IEC 61400-1 too. TurbSim will apply a wind field onto 200x200 grid points, such that the domain is greater than the rotor. In this way the turbine is not able to move outside of the wind field while experiencing a loading. Like the CFD Model, the undisturbed wind field has a turbulence intensity of 5%.

The wind field generation of TurbSim is used when to model full wake wind field. The only differences to the undisturbed wind field is the mean wind speed as well as the turbulence intensity. Both values are determined using Equation 1 and 2, as given in the IEC.

\[
U_{\text{wake}} = U_{\text{inf}} - \frac{1}{2} U_{\text{inf}} C_T \left( 1 + 0.17 \frac{x}{D} \right)^2 \tag{1}
\]

\[
Tl_{\text{wake}} = \sqrt{Tl_{\text{amb}}^2 + \left( \frac{1}{1.5 + 0.3 \frac{x}{D} \sqrt{U_{\text{inf}}}} \right)^2} \tag{2}
\]

Equation 1 represents the wind speed in the wake \(U_{\text{wake}}\) at distance \(x\), normalized by the Diameter \(D\) in relation to the undisturbed wind speed \(U_{\text{inf}}\) and to the thrust coefficient \(C_T\). Equation 2 determines the turbulence intensity inside a wake \(Tl_{\text{wake}}\) as a quadratic summation of the ambient turbulence intensity \(Tl_{\text{amb}}\) and the added turbulence.

Another wake situation investigated in this paper, occurs while the turbine is partially inside a wake. For this setup two wind fields will be merged along the mixing zone. However, the actual mixing of the flow field is neglected. While comparing the IEC flow field (Figure 5) with the CFD flow field (Figure 6), it becomes clear, that the results of the CFD environment lead to a clear distinction of free flow and wake flow. Inside the wake, the mean velocity is reduced more drastically. Meanwhile in the IEC flow field, the increased turbulence intensity can be clearly seen.

(a) Undisturbed
(b) Full wake
(c) Half wake

*Figure 5. Input flow fields according to IEC 61400-1*

(a) Full wake
(b) Half wake

*Figure 6. Input flow extracted out of the CFD environment*
3.3. Multi Body Simulation
The load calculation is done with the multibody software (MBS) Simpack. For the aerodynamic calculation the AERODYN code from NREL is used. AERODYN uses the dynamic stall model from Beddoes. Over the blade 17 aerodynamic profiles are equally distributed to determine the aerodynamic forces (see Figure 7a). The wind turbine model has flexible blades with 10 active eigenmodes and a flexible tower with 4 active eigenmodes. All other bodies are modelled as rigid.

The wind turbine model utilizes a standard PI-controller for the collective pitch system and below rated speed the generator follows a speed-torque characteristic with a hysteresis for the nominal torque, both control strategies are modelled by MATLAB/SIMULINK. The forces are calculated in the rotor hub coordinate system, given by the Germanischen Lloyd [26] (see Figure 7b).

4. Simulation Results
The following figures show the simulation results with the IEC-wind field and the CFD results compared to the reference wind field with a turbulence intensity of 5%. The mean wind speed in the free flow is at 11m/s.

The full wake means the turbine is directly in the wake field of the upstream sited wind turbine. The half wake plots have at the positive y-direction the wind field of the wake of the up-stream turbine, the other half of the field is the reference wind field.

4.1. IEC Wind field results
The IEC Wind field results are given in Figure 8. Figure 8a shows the time series of the thrust force on the rotor. The thrust force correlates directly with the mean wind speed and the pitch angle. The average wind speed of the field is near the rated wind speed of the wind turbine and therefore the thrust force is close to its maximum.

The full wake and the half wake wind field both have higher maximum thrust forces (up to 200 kN) than the reference and the amplitude is two times higher than the reference. The distribution of the thrust force can be seen in Figure 8b. The maximum distribution of the half wake is similar to the reference. The full wake indicates also a slight increase of the mean thrust force, meaning that under that loading condition, the turbine is pushed back further.

Figure 8c and 8d show the bending moment in the z-direction. The results for the IEC wind field are as expected. A higher turbulence intensity results in higher amplitudes of the forces and moments at the hub. It can be noticed that in half wake situation the maximum bending moment and the mean of the signal are below the reference. Nonetheless, the span of the half wake is wider than the reference signal, this can be traced back to an aerodynamic imbalance.
4.2. CFD Wind field results
The CFD wind fields are calculated as described in chapter 3. In comparison to the IEC wind fields the vortex from the upstream turbine is now part of the wind field and the wind speed in the downstream is lower than free wind field. This results in a higher turbulence, which is similar to the IEC approach and can be seen in Figure 9. The extremes of thrust force and bending moment in the z-direction experience an absolute increase in comparison to the reference (see Figure 9b and 9c). In this approach the half wake is even more critical because of the high bending moment and constant offset in the z-direction, due to the intensified aerodynamic imbalance between a high turbulent wind field with low wind speeds and a low turbulent field with high wind speed.
5. Discussion of Results

The simulations show that not only the full wake is relevant for wind turbines, the half wake or partial wake influence might be even more important to analyze. The comparison between the CFD Half Wake and the IEC half wake shows that the thrust, and the bending moment are comparable (see Figure 10a and 10c), whereas the bending moment in z-direction differs extremely (see Figure 10b).
These first results show that the wake effect needs to be considered in wind parks for different partial wake effects. The results also indicated that for special inflow situation, i.e. partial wake, the proposed methods are not sophisticated enough to be used to determine bending moments. As a next step various wake models will be compared for the use in load calculation, such that the wake inflow can be matched without the calculation demand. An additional step will be to take the complete lifetime of a wind turbine within a park into consideration to evaluate the difference in the load assumptions.

6. Conclusion
In the paper two various wake inflow models and their influence on the receiving wind turbine in a multi body environment were presented. As expected, it was seen that if the receiving wind turbine is situated completely inside the wake, the flow situation results in a greater fluctuation of the loads. The second inspected flow situation (Partial Wake) indicated a high fluctuation too. In addition, the flow situation also resulted in high bending moments, which is based on aerodynamic imbalance. This could lead to early fatigue. Therefore, it needs to be stated that the wake influence should not be neglected while designing a wind farm and the required wind turbine. In this paper, CFD was used to determine an improved inflow, which is not beneficial with respect to fast computation. In case of a partial inflow it is advised to use a better wake model then the one proposed by the IEC due to the fact that bending loads behave more as expected and lead to higher fluctuation. Such a fluctuation could lead to a premature fatigue error, if not considered during the design phase.

References
[1] Verwaltungsverfahren zur Festlegung des Höchstwerts für die Ausschreibungen für Windenergieanlagen an Land des Jahres 2018 nach §85a Absatz 1 Erneuerbare-Energie-Gesetz, Jochen Homann, 29.11.2017
[2] Internation Electrotechnical Commission standard DIN EN 61400-1, design requirements, Release 2015
[3] Werkmeister A, Schelenz R, Jacobs G, Calculation of the design loads with SIMPACK, Simpack, User Meeting, Conference, 09-2015, Augsburg
[4] Vermeer LJ, Sørensen JN and Crespo A 2003 Wind turbine wake aerodynamics Progress in Aerospace Sciences 39 467–510
[5] Sanderse B 2009 Aerodynamics of wind turbine wakes (Netherlands: Energy research Centre of the Netherlands) ECN-E-09-016
[6] Lissaman P 1979 Energy efficiencies of arbitrary of wind turbines J. Energy 3(6) 323-28
[7] Crespo A, Hernández J and Frandsen S 1999 Survey of modelling methods for wind turbine wakes and wind farms Wind Energy 2 1–24
[8] Vermenuilen P 1980 An experimental investigation of wind-turbines wakes 3rd Int. Symp. on wind energy systems (Copenhagen: Denmark) pp 214-23
[9] Milborrow D 1980 The performance of arrays of wind turbines J. Wind Eng. and Indus. Aero. 5 403-430
[10] Katie I, Højstrup J and Jensen N 1986 A simple model for cluster efficiency Conf. European wind energy (Rome: Italy) pp 407-10
[11] Voutsinas S, Rados K and Zervos A 1990 On the analysis of wake effects in wind parks J. Wind Engineering 14(4) 204-19
[12] Snell H 1998 Review of the present status of rotor aerodynamics J. Wind Energy 1 46–69
[13] Snell H 2003 Review of aerodynamics for wind turbines J. Wind Energy 6 203–211
[14] Sørensen J and Shen W 2002 Numerical modeling of wind turbine wakes J. Fluids Eng. 124(2) 393–99
[15] Mikkelseen R 2002 Actuator disc methods applied to wind turbines PhD thesis Technical University of Denmark
[16] Jimenez A, Crespo A, Migoya E and García J 2007 Advances in large-eddy simulation of a wind turbine wake J. Phys.: Conf. Ser. 75 1–13
[17] Shen W, Hansen L and Sørensen N 2009 Determination of the angles of attack on rotor blades J. Wind Energy 12(1) 91–8
[18] Shen W, Sørensen J, and Zhang J 2007 Actuator surface model for wind turbine flow computations Proc. Conf. European Wind Energy (Milan)
[19] Shen W, Zhang J, and Sørensen J 2009 The actuator-surface model: a new Navier-Stokes based model for rotor computations J. Sol. Energy Eng. 131(1) 011002
[20] Duque E, Dam C, and Hughes S 1999 Navier-Stokes simulations of the NREL combined experiment phase II rotor 37th Aerospace Sciences Meeting and Exhibit (Reno) 99-0037
[21] Johansen J, Sørensen N, Michelsen J, and Schreck S 2002 Detached-eddy simulation of flow around the NREL phase VI blade J. Wind Energy 5 185–97
[22] Zahle F and Sørensen N 2007 On the influence of far-wake resolution on wind turbine flow simulations J. Phys.: Conf. Ser. 75 012042
[23] Zahle F and Sørensen N 2008 Overset grid flow simulation on a modern wind turbine 26th Conf. Appl. Aero. (Honolulu)
[24] Jonkman J, Butterfield S, Musial W and Scott G 2009 Definition of a 5-MW reference wind turbine for offshore system development National Renewable Energy Laboratory TP-500-38060
[25] National Renewable Energy Laboratory, Hrsg. TurbSim user's guide: version 1.06.00. 2012.
[26] Guideline for the Certification of Wind Turbines, Germanischem Lloyd, 2010