

Simulation of a 5MW wind turbine in an atmospheric boundary layer

Konrad Meister, Thorsten Lutz and Ewald Krämer

Institut für Aerodynamik und Gasdynamik (IAG), Universität Stuttgart, Pfaffenwaldring 21,
70569 Stuttgart, Germany

E-mail: meister@iag.uni-stuttgart.de and lutz@iag.uni-stuttgart.de and
kraemer@iag.uni-stuttgart.de

Abstract. This article presents detached eddy simulation (DES) results of a 5MW wind turbine in an unsteady atmospheric boundary layer. The evaluation performed in this article focuses on turbine blade loads as well as on the influence of atmospheric turbulence and tower on blade loads. Therefore, the turbulence transport of the atmospheric boundary layer to the turbine position is analyzed. To determine the influence of atmospheric turbulence on wind turbines the blade load spectrum is evaluated and compared to wind turbine simulation results with uniform inflow. Moreover, the influences of different frequency regimes and the tower on the blade loads are discussed. Finally, the normal force coefficient spectrum is analyzed at three different radial positions and the influence of tower and atmospheric turbulence is shown.

1. Introduction

Modern wind turbines are higher than 100m and have rotor diameters up to 120m and more. They play an important role in clean power production and compete against other power production methods like solar-, hydro-, nuclear-, gas- and coal-power, just to mention a few. Therefore, reliability and economic feasibility are the most important factors in wind power production. The latter is achieved with increasing size of the turbines, maximum power output and choosing operation areas with constant high wind speeds. These places are often found offshore, where reliability is a major criterion as maintenance is expensive and not available at any time. To afford effective working wind turbines it is essential to know the loads they are exposed to. As wind turbines are operating in the atmospheric boundary layer they are affected by unsteady wind conditions like turbulence, gusts and shear [1]. These inflow conditions lead to a change in wind speed and angle of attack over time and height. It is known from literature that both variations lead to unsteady, nonlinear blade load response even in the linear region of originally steady aerodynamic polars. The effects caused by changing angle of attack are well described by Theodorsen's theory [2]. Experimental results of Theodorsen's theory on oscillating wing sections can be found in [3]. The change of velocity due to gusts is known as the Sears's problem [4]. Besides these various other phenomena exist, which are well described in [5]. Due to the complexity of the topics a various number of simulation methods exists focusing on different priorities. [6] provides a convenient categorization of the different methods for wind turbine simulations. They range from simple Blade Element Momentum (BEM) theory based models [7] over actuator disc/line models [8, 9] up to so called direct models [10, 11, 12, 13] and differ in computational effort and accuracy. As the BEM method is based on calculating



two dimensional load distributions along the blade by using tabulated airfoil data it needs corrections for root and tip losses, 3D effects, dynamic stall models and models for wake and inflow dynamics. The advantage is that the BEM is very fast and can be easily implemented e.g. in aeroelastic codes [14]. The actuator disk method is based on performing a CFD simulation where the turbine is modeled by an actuator disk imposing a body force on the flow field. Therefore, the actuator disk uses tabulated airfoil data like the BEM, but does not need root or tip loss corrections and wake or inflow models as these effects are taken into account by the Navier-Stokes-Equations. The actuator line method is an extension of the disk model, where the body forces are non-uniformly distributed in the rotor plane and thus the blade position is time dependent. The so called direct model is the most expensive model. It is based on simulating the flow around a discretized blade in 3D and is therefore able to consider all effects from the actuator line plus 3D viscous effects, dynamic stall, interaction with atmospheric turbulence and many more without need for modeling.

To focus on the associated unsteady effects the IAG performs detailed CFD simulations of wind turbines under atmospheric inflow conditions. The simulations are using a direct model, which means that the whole aerodynamic shape of the turbine including blades, tower and nacelle is considered geometrically correct by using friction walls. This article presents simulation results of a 5MW wind turbine sited in the German offshore test field Alpha Ventus which were performed in the German OWEA project.

2. Numerical approach

To analyze the aerodynamic behavior of wind turbines in operation RANS based CFD simulations are performed at the IAG by using a process chain for wind turbine simulations [11]. The two major parts of this process chain are the commercial grid generator Gridgen by Pointwise which supports scripting capability for automated meshing and the block structured RANS based flow solver FLOWer which is provided by the German Aerospace Center (DLR) [15]. To consider atmospheric turbulent inflow in wind turbine simulations FLOWer was extended by an unsteady Dirichlet boundary condition. This boundary condition uses measured, artificial or simulated, steady or unsteady, ASCII or BINARY data as inflow data and feeds it into the flow simulation upstream of the turbine. To fit the discretization of the simulation in space and time the inflow data are interpolated to the computational mesh by using a bilinear interpolation, while for the time a linear interpolation is used.

2.1. The turbine

The simulations were performed on the 5M, a 5MW offshore wind turbine being one of two wind turbine types which are sited in the German offshore test field Alpha Ventus. The 5M has a rotor diameter of 126 m and a hub height of 95 m. The rotor plane is tilted by 6° and the blades provide a precone of -4° . The chosen rotor frequency is 12 rpm. To analyze if unsteady aerodynamic effects occur at this frequency the reduced frequency k needs to be calculated. Here, k is defined by

$$k = \frac{\omega \cdot c}{2V}, \quad (1)$$

where the angular blade frequency $\omega = 2 \cdot \pi \cdot 0.2 \text{ Hz}$ and c is the chord. The flow velocity $V = \sqrt{U_\infty^2 + (\omega \cdot r)^2}$ is calculated with the local blade radius r . Due to the change of chord and flow, k is a function of r . Here k will be analyzed at three different radial positions which are located at inboard region ($r_i = 11.5 \text{ m}$) mid region ($r_m = 31.5 \text{ m}$) and tip region ($r_t = 59.5 \text{ m}$). Using the estimation $U_\infty = 15 \text{ m/s}$ the reduced frequencies can be calculated (table 1). Due to the reduced frequency distribution unsteady nonlinear aerodynamic effects can be expected for blade radius positions lower than 31.5 m and should be considered. However, it has to be

mentioned that the reduced frequencies listed in table 1 do not depend on the inflow situation which might have an additional influence on unsteady aerodynamic effects.

The performed CFD simulations in this article are able to consider unsteady nonlinear aerodynamic effects, as the blade forces are calculated directly from the current unsteady flow conditions. To perform a simulation of the 5M, the turbine was splitted into several parts to create block structured meshes. In total the complete turbine CFD model consists of different meshes describing tower (including the nacelle), spinner, background and blades, plus several auxiliary meshes. All these meshes are combined in FLOWer by using the CHIMERA technique for overlapping grids [16].

The blade meshes were created by using a dedicated blade mesh generation script which was developed at the IAG [17]. This script uses the blade surface in CAD data “.iges” format as basis and automatically creates a complete blade mesh. To influence the blade mesh resolution several parameters like cell distribution, cell numbers, growth rate factors and many more need to be specified. In doing so a blade mesh with approx. 1.9 million cells and a fully resolved boundary layer with $y^+ \approx 1$ for the first boundary layer cells was created. The background mesh was created manually with a total size of $640\text{ m} \times 560\text{ m} \times 404\text{ m}$. It was designed with a grid resolution of 2m in horizontal and downstream direction. In vertical direction the cell spacing increases from 10^{-5} m at ground level up to 2m with a constant growth rate of approx. 14%. This was needed to resolve the lower boundary layer of the later performed DES which was simulated in URANS mode. The vertical spacing of 2m is constant up to a height of 280 m. Above this point the cell spacing is increasing constantly to save grid points until the total height of the background mesh of 404m is reached. The other meshes were created manually, too. Their walls were defined as friction walls with an approximated design value of $y^+ \approx 1$ for the wall nearest cell layer. Normal to the wall the cell size is increased by a constant growth ratio which is lower than 20% for each mesh. A brief overview of all meshes used in the turbine model is given in table 2, while a schematic view of the computational domain is given in figure 1, where the inflow is at $x = 0\text{ m}$ and the turbine is positioned at $p_T = (320, 0, 0)$. In total the complete computational setup consists of approx. 35.5 million cells.

Table 1. Reduced frequency for three different cuts.

r [m]	k
11.5	0.130
31.5	0.051
59.5	0.009

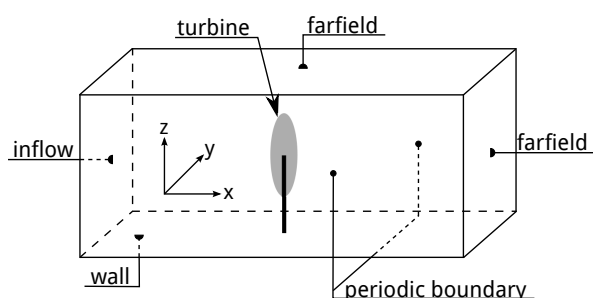


Figure 1. Schematic view of the computational domain. The turbine is positioned in the center of the computational domain 320 m downstream the inflow plane.

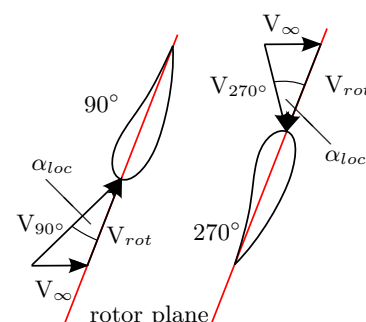


Figure 2. Change of local angle of attack α_{loc} at 90° and 270° azimuth due to rotor tilt angle. Red line defines tilted rotor plane. Left blade is going down, right is going up.

Table 2. Overview of meshes used for the simulation of the 5M

Mesh	Cells (mio)	No.	Description
Background	23.8	1	Steady mesh covering the environment around the turbine.
Blade	1.9	3	Rotating mesh with fully resolved boundary layer $y^+ \approx 1$
Hub	0.8	1	Rotating mesh with fully resolved boundary layer.
Tower	3.5	1	Steady mesh with fully resolved boundary layer covering tower and nacelle.
Auxiliary	1.7	-	Several smaller rotating and steady meshes building connections to other meshes

2.2. Computational set up

To study the influence of turbulence on rotor loads, two different turbine simulation cases were performed using the previously described meshes. Case I uses a uniform wind speed over time and height while case II feeds in data of an unsteady turbulent atmospheric boundary layer. The two performed cases are identical in their computational setup except the inflow far upstream (see table 3) which is described in the next section. Both cases were detached eddy simulations (DES 97) with a Spalart Almaras turbulence model. The time step of the simulation was 0.02083 seconds which is identical with 1.5° azimuthal movement. A maximum of 30 inner iterations was performed during each time step. In both cases more than 35 full revolutions were performed. The simulations were performed using rigid blades. Pitch and rotor frequencies were constant. The simulations were taken out on 72 AMD Opteron(tm) 6276 processors with 16 cores each on the High Performance Computing Center Stuttgart. The computational time for both simulations was approx. 300 hours.

Table 3. Some simulation parameters for case I and II.

	Case I / II
Inflow	15.0 m/s / turb.
Time step	1.5°
Rotor freq.	0.2 Hz
Tilt	6.0°
Precone	4.0°
Pitch	11.2°

2.3. Inflow data

The inflow data for the case II simulations were extracted from a LES simulation of an atmospheric boundary layer performed at ForWind (Center for Wind Energy Research) in Oldenburg, Germany. This simulation was taken out during the German OWEA project using the flow solver PALM [18]. The simulation covered an area of $1000 \text{ m} \times 1000 \text{ m} \times 900 \text{ m}$ in xyz direction and was resolved with a time step of 0.25 seconds. From this simulation flow data at x-constant planes were extracted with a temporal resolution of 0.25 seconds for over 10 minutes. These data define the basis for the later used inflow.

Very often in wind turbine simulations a periodic inflow dataset is desired, to control whether the wind turbine aerodynamics converge in time, when feeding in the same inflow data again and again. Otherwise it would not be possible to determine if time depended fluctuations are caused by bad convergence or by the flow physics. Unfortunately the time steps in wind turbine simulations are much smaller than 0.25 seconds and a full 10 minute time series would be too expensive to simulate especially when the dataset has to be fed in more than once. Therefore, a shorter 60 seconds long periodic time signal was created from the 10 minute LES data set. This was achieved by extracting a 70 seconds time signal out of the basic signal for every point (figure 3(a)) and combining the first and last 10 seconds to a new 10 seconds time signal by using equation 2. This new time signal $f_r(t)$ is added in front of the inner signal $f_i(t)$ to receive a smooth 60 seconds periodic signal (figure 3(b)). It can be seen that the result is periodic

in time, and that the used method is mathematically well defined. However, the weakness by cutting a subset from a long time series is, that the result mainly depends on the data set being chosen. Moreover, according to the short time of the modified dataset it does not provide the correct statistics of the atmospheric turbulence. Nevertheless, for the analysis of unsteady loads and effects on the wind rotor caused by turbulence, without focusing on atmospheric statistics, this method seems adequate. It should be mentioned, that in the future, when computational power is increasing, it will be possible to simulate larger time series up to several minutes.

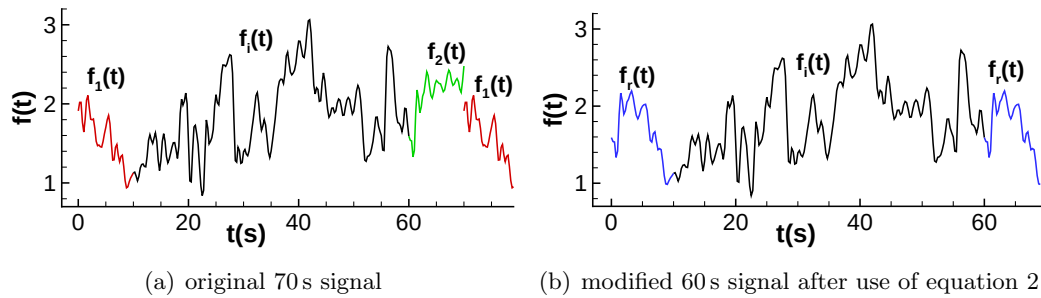


Figure 3. Time signals before and after use of equation 2. Same colors denote same signal.

$$f_r(t) = \frac{1}{2} \cdot \left[f_1(t) \cdot \left(1 - \cos \left(\frac{t}{10} \pi \right) \right) + f_2(t) \cdot \left(1 + \cos \left(\frac{t}{10} \pi \right) \right) \right] \quad (2)$$

Analyzing the inflow data a strong shear in horizontal direction could be observed. Therefore, it was decided to use periodic boundary conditions on both sides of the background mesh (see figure 1). Unfortunately, the inflow data was not periodic in horizontal direction for the chosen span. For this reason, a final horizontal periodic signal was created from the 60 seconds periodic time series. This was achieved by using equation 2 for the spanwise direction, too. These final data define the inflow conditions for the later performed case II simulation.

3. Simulation results

The objective of the performed simulations was to receive information on how the wind turbine loads are effected by atmospheric turbulence, gusts and shear. Therefore, two cases one with uniform inflow and the other one with turbulent atmospheric boundary layer inflow have been analyzed.

3.1. Uniform inflow

Due to the uniform inflow periodic loads are expected on the rotor. The load evaluation of the rotor blades is shown in figure 4. Plotted is the normalized axial blade force for each blade over its own azimuth position. It is noticeable that all three blades show the same load behavior over one full rotation. Moreover, a significant load reduction of 22% in front of the tower at 180° azimuth can be seen followed by a quick load increase of 22% over the next 30° of azimuth. This shows that the tower has a distinct influence on the aerodynamics. As the tower influence can be expected between 120° and 240° the reduced frequency calculated from the tower is three times higher, than for the rotor, which means that the tower increases the need of considering unsteady aerodynamic effects. Besides the tower influence a shift of the loads to the left side can be observed, even though the inflow is uniform. This is caused by the rotor tilt which changes the angle of attack permanent over one rotation. Analyzing the local angle of attack α_{loc} at 90° and 270° azimuth shows that α_{loc} is higher for 270° position than for 90° position (figure 2). On the other hand the relative velocity is larger at 90°. Due to that the loads can either shift to

the left or the right side, depending on which effect is dominating, the change in angle of attack or the change in relative velocity.

To analyze the load distribution along the blade, the blade normal force coefficient spectrum $\bar{C}'_n$ is depicted in figure 5 for the three positions analyzed in table 1. The three spectra show load peaks at the blade passing frequency of 0.2 Hz and their higher harmonics, which fits very well to the load reduction in figure 4 at 180°. The strength of $\bar{C}'_n$ is similar at these frequencies till $f = 3$ Hz. Above this the r_i and r_m spectra show a slight increase of unsteady load fluctuations caused by the aerodynamics.

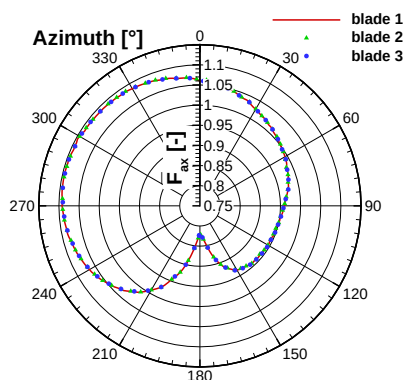


Figure 4. Normalized axial blade force $\bar{F}_{ax}$ for all three blades over blade own azimuth. Normalized with mean axial blade force of a Hann window.

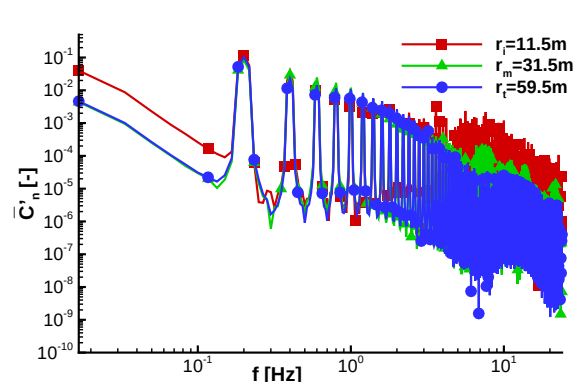


Figure 5. Spectrum of chord normal force coefficient $\bar{C}'_n$ evaluated over 60 seconds using Normalized with mean chord normal force coefficient.

3.2. The pure atmospheric boundary layer inflow

Before running the simulation of case II an additional pure simulation (DES 97) of the atmospheric boundary layer inflow without wind turbine was performed. This simulation was used to determine how turbulence develops and to analyze which frequency regimes can be resolved. For the simulation a shorter version of the turbine background mesh was used ($0 \text{ m} \leq x \leq 448 \text{ m}$). Apart from that the background mesh was identical. Moreover, the numerical setup is identical to the turbine setup (DES 97, SA turbulence model). Figure 6 shows the turbulence intensity distribution for several downstream positions at $z = 95 \text{ m}$ height (hub height) over y -direction for the pure boundary layer simulation. The shift due to the vertical velocity component is neglected in this evaluation method, as it is very small. The first three lines from top to bottom describe the distribution at the inflow plane, 64 m further downstream and at 320 m downstream, which is the turbine distance in the rotor simulations. The fourth line is extracted at 320 m, too, but calculated with another numerical scheme using higher order methods to reduce numerical dissipation.

The first three lines show that most of the decrease of turbulence intensity happens over the first 64 m and then stays nearly constant up to $x = 320 \text{ m}$ which is the distance between inflow and rotor in the turbine simulations. Only at $y = -220 \text{ m}$ this cannot be observed. The highest value is reached at $y \approx 100 \text{ m}$. The fourth (green) line of figure 6 is additional and was extracted from a simulation of the pure atmospheric boundary layer using a 5th order WENO scheme [19], which was implemented in FLOWer at the IAG [20]. Even though WENO was not used for the turbine simulations it provides interesting information about the turbulence transport. On balance, WENO predicts higher turbulence intensity and a less smoothed distribution. Compared to the standard linear scheme the loss with WENO is lower.

Evaluating the transport of the velocity spectrum, provides important information about the

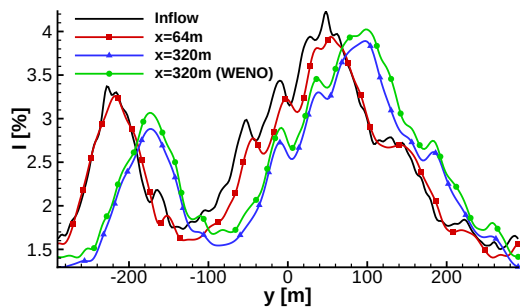


Figure 6. Turbulence intensity distribution at a constant height of $z = 95$ m (approx. rotor center) at several positions downstream from the inflow. Picture is made from behind, looking upstream.

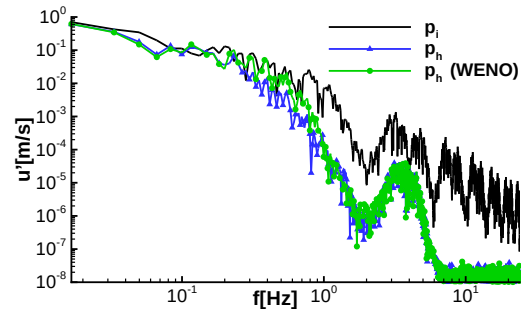


Figure 7. Spectrum of main velocity fluctuations at a constant height of $z = 95$ m (hub height). Black line is extracted at inflow, blue and green line at 320 m downstream.

dissipation the code causes by numerical scheme, discretization and time step. To analyze the fluctuation transport, at least two points at different downstream positions are needed which are connected by flow direction. For the current simulation $p_h(x, y, z) = (320, 0, 95)$ (position of hub of the performed turbine simulations) and its corresponding point at the inflow $p_i(x, y, z) = (0, -48, 95)$ are chosen. Both points are connected by the mean flow direction which has a shift of $\Delta y = +48$ m over $\Delta x = +320$ m (see figure 6). Figure 7 shows the x-velocity fluctuation spectrum of the boundary layer simulation at p_i (black line), at p_h with standard FLOWer (blue line) and at the same position calculated with WENO (green line). For low frequencies up to 0.2 Hz the amplitudes of all lines are of similar order. Above 0.2 Hz the standard scheme begins to underestimate the inflow spectrum, while the WENO scheme is still adequate till 0.5 Hz. The descend of WENO above 0.5 Hz is stronger than for the standard scheme, so that both curves show the same trend above 1 Hz. At 2 Hz the three curves show a local minimum. This is exact the frequency which is resolved from the time step of the ForWind LES simulation. The higher frequency regime $f > 2$ Hz shows an increase of the amplitudes. These arise because of the linear time interpolation of the data at the inflow plane. To avoid this artificial increase in future, it is planned to use fourier based functions for the time interpolation.

3.3. Turbine under atmospheric turbulence inflow

Figure 8(a) shows the load distribution of the blades over 360° azimuth for the turbulent inflow. Caused by turbulence and the variation of inflow the loads change permanently. The maximum of $\bar{F}_{ax}$ is reached blade depending between 280° and 300°. At 300° $\bar{F}_{ax}$ decreases slightly for all three blades. Looking at blade 2 at 120° and blade 3 at 240° a jump in the force distribution can be observed. This jump occurs as the flow conditions change over one full revolution. Moreover, the force distribution shows the tower influence at 180° and the asymmetry due to the rotor plane tilt angle which could already been detected for the steady inflow conditions. Analyzing $\bar{F}_{ax}$ of blade one over one inflow period (see figure 8(b)) shows the influence of the complete atmospheric turbulent inflow sequence. The axial blade force changes permanently over time. Most times $\bar{F}_{ax}$ varies between 0.9 and 1.1, if the tower influence region is neglected. In front of the tower an increase of 30% between the smallest and the highest loads can be found. The highest load is found at 240°, where $\bar{F}_{ax}$ is increased by almost 25%. In addition to the axial loads the normalized power distribution over 60 s is shown in figure 9. It can be seen, that the power has significant fluctuations like the axial force.

To analyze the load fluctuations from figure 8(b) and for comparison to the uniform inflow load case figure 10 contains the spectrum of $\bar{F}_{ax}$ for both cases evaluated over 60 seconds. Both

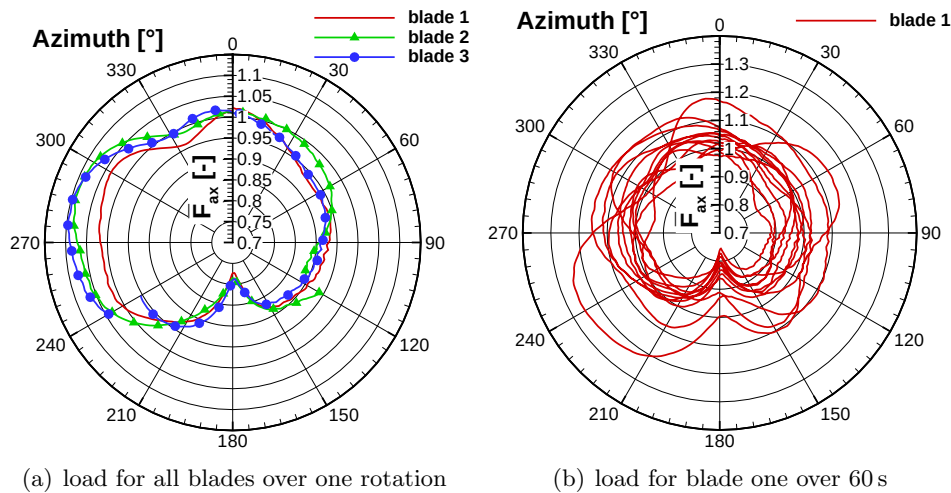


Figure 8. Axial normalized blade force $\bar{F}_{ax}$ of case II simulation. Normalized with mean axial blade force of case II simulation.

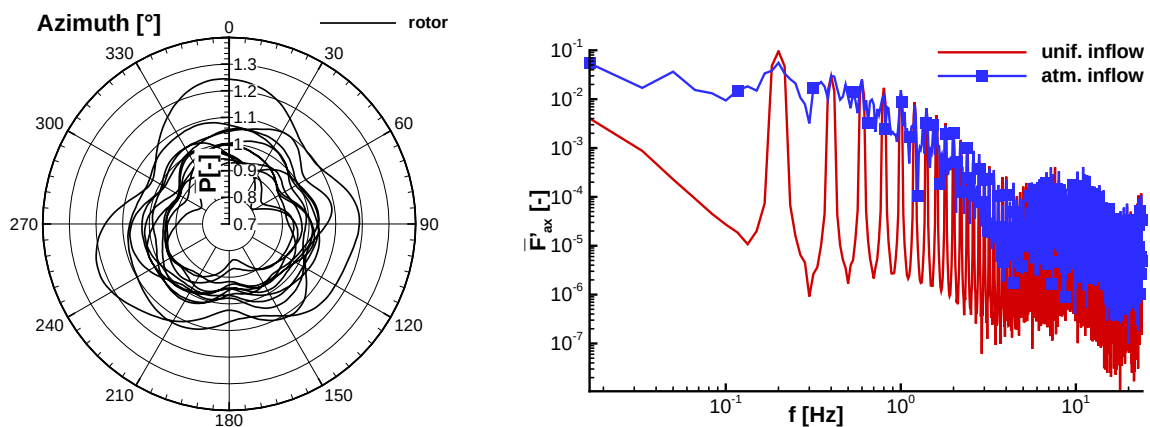


Figure 9. Normalized rotor power $\bar{P}$ of case II over 12 rotations (60 sec). Normalized with mean power of case II. **Figure 10.** Spectrum of $\bar{F}_{ax}$ evaluated over 60 seconds for both turbine simulations using a Hann window.

curves predict the amplitudes for the blade passing frequency (0.2 Hz) and their higher harmonics. However, the tower influence is smaller in case II simulations (-36% at 0.2 Hz). This is caused by the boundary layer which, compared to uniform inflow, has lower wind speeds at low rotor height. Apart from that, due to the atmospheric turbulence, the case II spectrum is higher than the case I spectrum. The highest atmospheric amplitude appears up to a frequency of 0.2 Hz. Smaller amplitudes of 1% and higher appear until 0.7 Hz. Above that point the spectrum decreases faster. Nevertheless, even in the high frequency range the load spectrum of the atmospheric turbine simulation predicts higher load fluctuations.

To analyze how the higher frequencies affect the blade loads a back-transformation of the load spectrum to the time signal is performed. Figure 11(a) & (b) contain three different time signals with and without the excited blade tower induced frequencies. Each signal was created using a back-transformation beginning at the lowest frequency of 0 Hz up to the a defined higher maximum frequency. The amplitudes lying above this upper limit are not considered in the specific time signal. The 25 Hz signal is the most accurate time signal as it contains all frequencies. As can be seen from figure 11(a) the 1.5 Hz curve predicts the accurate signal much better than the

1 Hz. The highest deviations of the 1 Hz reconstructed time signal can be found in the tower region. Moreover, smaller deviations exist at various azimuth positions like 100° , 135° , 210° , 350° . To exclude the influence of the blade tower interaction and their higher harmonics, figure 11(b) shows the reconstruction without these frequencies. Without tower influence the 1 Hz and 1.5 Hz signal predict the 25 Hz much better, especially in the tower region. The measured maximum deviation to the 25 Hz signal of figure 11(b) was approx. $\pm 2\%$ for the 1 Hz and $\pm 1.2\%$ for the 1.5 Hz signal. It is expected that these deviations increase, if the higher frequency regime can be transported with less dissipation.

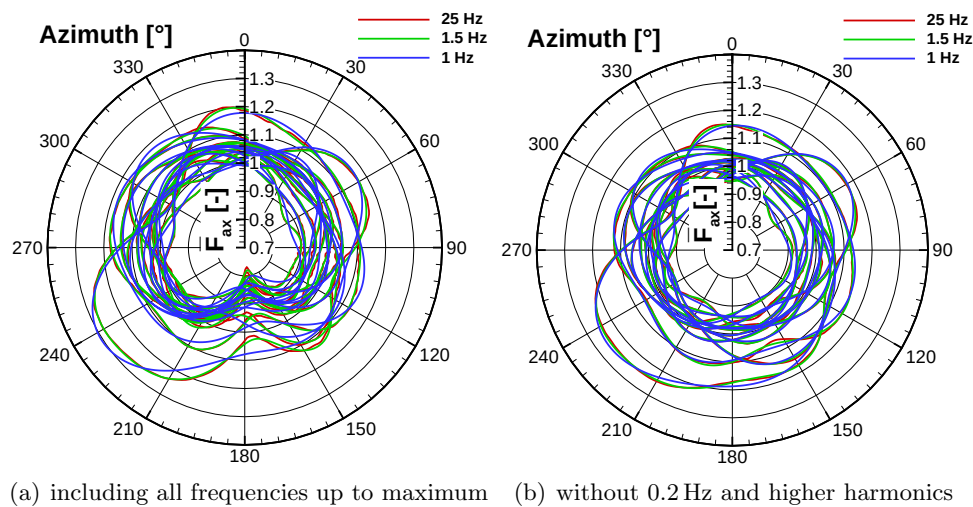


Figure 11. Axial blade force $\bar{F}_{ax}$ received from a back-transformation of axial blade force spectrum considering frequencies between 0 Hz up to a maximum of 25 Hz, 1.5 Hz or 1 Hz.

To analyze the load distribution along the blade, figure 12 shows the spectrum of the normalized chord normal force coefficient $\bar{C}_n$ for the three positions which were evaluated in table 1. Up to the blade passing frequency the two innermost spectra are of similar order, while the outermost spectrum shows lower amplitudes. As the latter cut is close to the tip it seems to be obvious that lower loading occurs. r_i and r_t show a distinct amplitude at 0.2 Hz while for r_m it is in the range of the surrounding amplitudes. Going to higher frequencies the innermost curve drops fastest, while r_m is in the middle. This trend changes at 3 Hz, the identical frequency like in figure 5, when the amplitudes of r_i and r_m recover while the outermost curve is still dropping.

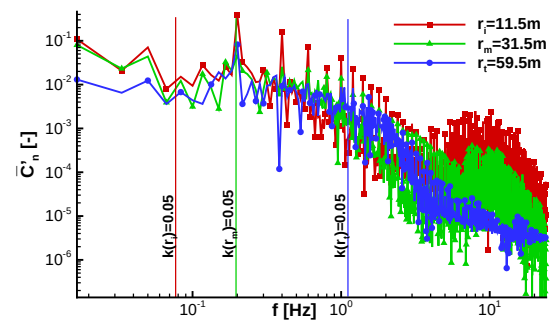


Figure 12. Spectrum of chord normal force coefficient $\bar{C}'_n$ evaluated over 60 seconds (inflow period). Normalized with mean chord normal force coefficient.

To give an impression in which frequency regime unsteady aerodynamic effects might occur, three frequency lines are included in figure 12 marking the reduced frequency of $k = 0.05$ for each of the analyzed blade sections. If this frequency lines are compared to the inflow spectrum (figure 7) it becomes clear that, due to the inflow turbulence additional strong unsteady aerodynamic effects might occur until have the blade span, especially when WENO would have been used, while the tip region seems not to be affected tremendously by unsteady aerodynamic influence from the

inflow. Nevertheless, the spectrum in figure 12 shows, that even close to the tip strong variations of $\bar{C}_n$ exist in the regime of $k \approx 0.05$.

4. Conclusions

This article analyzed the influence of atmospheric boundary layer turbulence on wind turbine loads by using a full rotor CFD model. Therefore, a 60 seconds periodic time series of LES data was fed into a detached eddy simulation of a wind turbine. It could be shown that large scale turbulence can be well transported to the downstream turbine, but smaller turbulence at higher frequencies dissipates faster. However, it was presented that higher numerical schemes can reduce turbulence dissipation. To demonstrate the influence of a turbulent atmospheric boundary layer the loads were compared to a wind turbine simulation with uniform inflow. It could be shown that the blade loads in turbulent atmospheric boundary layer increase compared to the loads in uniform inflow. Besides this, the influence of the tower on the blade loads was depicted. Moreover, an analysis of the load spectrum showed the influence of high frequencies on the blade loading with and without tower effects. In addition to this, it was received that load amplitudes higher 1 Hz still influence the loading. Finally, an evaluation of the normal force coefficient C_n at three radial positions showed the influence of turbulence along the blade. This evaluation was extended by calculating the frequency of the three chosen radial positions where unsteady aerodynamic effects might occur. These frequencies were then used for a closer view on the inflow data and the spectrum of C_n .

Acknowledgments

The presented studies were part of the RAVE project “OWEA”. The authors wish to acknowledge the “German Federal Ministry for the Environment, Nature Conservation and Nuclear Safety” for the funding of this research. The computational resources for the simulations were kindly provided by the “High Performance Computing Center Stuttgart”.

- [1] Mücke T, Kleinhans D and Peinke J 2011 *Wind Energy* **14** 301–316
- [2] Theodorsen T 1935 General theory of aerodynamic instability and the mechanism of flutter Tech. rep. NACA Report 496
- [3] Piziali R A 1994 2-D and 3-D Oscillating Wing Aerodynamics for a Range of Angels of Attack Including Stall Tech. rep. Ames Research Center - NASA
- [4] von Kármán T and Sears W R 1938 *J. Aeronaut. Sci.* **5** 379–90
- [5] Leishman J G 2002 *ASME Conference Proceedings* **2002** 141–167
- [6] Sanderse B, van der Pijl S and Koren B 2011 *Wind Energy* **14** 799–819
- [7] Jeon M, Lee S and Lee S 2013 *Renewable Energy* 1–6
- [8] Troldborg N, Sørensen J and Mikkelsen R 2007 *Journal of Physics* **75**
- [9] Troldborg N 2008 *Actuator Line Modeling of Wind Turbine Wakes* Ph.D. thesis Technical University of Denmark
- [10] Laursen J, Enevoldsen P and Hjort S 2007 *Journal of Physics* **75**
- [11] Meister K, Lutz T and Krämer E 2010 *Proceedings: German Wind Energy Conference, DEWEK*
- [12] Bechmann A, Sørensen N N and Zahle F 2011 *Wind Energy* **14** 677–689
- [13] Meister K, Lutz T and Krämer E Oldenburg, February 22–24, 2012 *EUROMECH[528] - Wind Energy and the impact of turbulence on the conversion process*
- [14] Jonkman J M and Buhl M L 2007 *WindPower 2007 Conference & Exhibition*
- [15] Kroll N and Fassbender J ISBN 3-540-24383-6 *Springer Verlag Berlin/Heidelberg/NewYork*
- [16] Chesshire G and Henshaw W 1990 *Journal of Computational Physics* **90**(1) 90 1–64
- [17] Meister K, Lutz T and Krämer E Madrid, October 20–22, 2009 *EUROMECH[508] - Wind turbine wakes*
- [18] Raasch S and Schröter M 2001 *Meteorol. Z.* **10** 363–372
- [19] Shu C W Essentially Non-Oscillatory and Weighted Essentially Non-Oscillatory Schemes for Hyperbolic Conservation Laws Tech. rep. Institute for Computer Applications in Science and Engineering, NASA Langley Research Center
- [20] Flad D 2011 *Implementation of a higher order method for flux calculation in FLOWer* Thesis Institute of Aerodynamics and Gas Dynamics (University of Stuttgart)