Investigation of the validity of BEM for simulation of wind turbines in complex load cases and comparison with experiment and CFD

H. Rahimi¹, B. Dose¹, B. Stoevesandt ², J. Peinke¹,²
¹ ForWind, Institute of Physics, University of Oldenburg, Küpkersweg 70, 26129 Oldenburg, Germany
² Fraunhofer IWES, Küpkersweg 70, 26129 Oldenburg, Germany
E-mail: hamid.rahimi@uni-oldenburg.de

Abstract.
The aim of this work is to investigate the validity of simulation codes based on the Blade Element Momentum (BEM) theory for three important design load conditions. This paper includes the cases of yawed inflow, rotor tower interaction for downwind turbines and the standstill case. Computational Fluid Dynamics (CFD) and experimental data (when available) are used for the evaluation of the obtained results. For the yawed inflow, the results indicate that significant deviations between BEM and experiments & CFD can be observed. This discrepancy is caused by unsteady phenomena such as the advancing & retreating blade effect and the skewed wake effect. In the case of the rotor and tower interaction of the downwind turbine, the results show that the BEM based code overpredicts the sectional forces in terms of the normal and tangential forces by 20 %. In the case of standstill, the evaluation of the results based on tip deflections shows clear differences in the output of both numerical approaches. While the flapwise deflections show a reasonable agreement, the CFD-based coupled solver predicts much larger edgewise vibrations.

1. Introduction
One of the most common approaches to simulate the aerodynamics of wind turbines is the Blade Element Momentum (BEM) theory. Low costs of the simulations, which are a result of the two dimensional theory, make the simulation of more than thousand load cases within a short amount of time affordable. Nevertheless, complex three dimensional flow phenomena which can occur during the operation of wind turbines cannot be captured by the basic BEM theory and have to be included by correction models for the specific phenomena.

Several experimental and numerical studies showed that simulation codes based on the BEM theory are in certain cases not sufficiently accurate and reliable for predicting the complex aerodynamic behavior of wind turbine blades [1, 2]. Some of the complex cases which can arise are: The aerodynamic behavior of the rotor blades under yawed conditions, the interaction of the rotor and the tower for downwind turbines, and the interaction of the blade structure with the incoming wind during standstill.

This work investigates the validity of Blade Element Momentum (BEM) based simulation codes for the listed conditions. Computational Fluid Dynamics (CFD) and experimental data
(when available) will be used for the purpose of the evaluation of the obtained results. Key aerodynamic quantities such as power, thrust and sectional forces are investigated using BEM for different turbine sizes and compared to wind tunnel measurements and CFD results. Based on that, the capabilities of BEM for the simulation of such complex conditions are discussed in this paper.

2. Approach
In this work, the wind turbine simulation code FAST [3] is used for the BEM calculations. The CFD simulations are performed using the open-source software OpenFOAM [4]. In case of the NREL Phase VI rotor, experimental results were included in the investigations. For large wind turbines, however, there is a lack of available measurement data. Therefore, only numerical results obtained by CFD are used to evaluate the validity of BEM in those cases. In the following sections, three complex scenarios are presented and the results are discussed.

3. Yawed flow
One of the common difficulties can arise during the modeling of the aerodynamic behavior of rotor blades under yawed conditions. These cases can result in different problematic situations such as: flow separation, azimuthal variation of the loads and an unbalance in the flow induced by the skewed wake [1, 5]. Thus, currently a major concern regarding BEM methods is the applicability of yaw models in the design process of wind turbines. The yaw models used are usually suffering from uncertainties such as the general validity of the model at extreme yaw angles and dynamic yaw effects[5, 2]. Therefore, it is essential to conduct more complex and realistic simulations in order to capture all the important details, which play a role in the correct modeling of wind turbines.

In this part of the work, the NREL VI wind turbine [6] is simulated using CFD and FAST. In CFD the simulations are conducted using the Spalart-Allmaras-DDES model [7] for 30 degree of yaw and a wind tunnel speed of 5m/s. The computational grid has a spherical shape and is fully structured and based on hexahedrons. The total mesh size accounts to 22 million cells. It features 300 cells in chord wise direction around the airfoils, 250 cells in the span-wise direction for each blade, and 300 cells in the wall normal direction. The $y^+$ values at the surface are kept below one everywhere on the blade surface. Fig. 1 shows a representation of the grid at the mid span section for the leading edge, trailing edge, and half of the complete domain.

Different grids were created and two methods were used in order to examine the grid-independent behavior of the solution, namely the systematic grid refinement and the Grid Convergence Index (GCI). The results of the study are discussed in [8].

In order to solve the pressure-velocity coupling, the PIMPLE algorithm which is a combination of the loop structures of SIMPLE [9] and PISO [10] is used. The numerical simulations are performed using the Facility for Large-Scale cOmputations in Wind energy research (FLOW) at the University of Oldenburg. In total 264 computational cores are used for approximately 96 hours. The effects of the tower and the nacelle are neglected in the performed simulations. The physical time step is set to $\Delta t = 2 \times 10^{-4}$s. Numerical convergence is achieved after 5 rotor rotations.

For the FAST simulations, three different yaw correction models, namely the model of Pitt and Peters [11], the model from Schepers [5] and the Generalized Dynamic Wake (GDW) model [11] are used. In Fig. 2 the CFD results show a reasonable agreement with the experimental data for both normal and tangential force coefficients. It should be also mentioned that the simulation results presented in [8] and [12] show a good agreement for the case of 0 degree of yaw.
In terms of the normal force coefficient at 30% span, all three correction models are predicting the maximum forces at the same azimuthal position. The maximum is reached in the downwind side of the blade. The model by Pitt & Peters shows a smaller amplitude and predicts a pure sinusoidal behavior. This can be due to the fact that Pitt & Peters model is not taking the root free vortex effect into account. The GDW model is underestimating the forces at the upper half of the rotor blade significantly. The model from Schepers has a closer agreement to the experimental results in terms of amplitude and phase. The result for the tangential force coefficient using these three correction models shows a higher deviation from the experimental results than the CFD calculations. All the correction model results are shifted towards the downwind side of the blade. The model from Schepers has a closer agreement to the experiments since this model is considering the root free vortex effect as a function of blade radius. Moreover comparing the advancing and retreating blade effect with the skewed wake effect, it can be concluded that in this section of the blade, both effects contribute to the load variation. This can be seen from the fact that the maximum forces are shifted towards the downwind side of the blade.
At the section of 95% of the span, both the normal and the tangential force coefficient of the experimental data and CFD results are showing a sinusoidal behavior which is induced by the tip vortex. The Pitt & Peters model is predicting a different phase and amplitude and the GDW model is predicting the correct phase but not the amplitude. The Schepers model also miss-predicts the position of maximum force and under predicts in average the forces. This could be due to the fact that these models are neglecting the effect of the tip vorticity in yawed flow in general.

Comparing the advancing and retreating blade effect with the skewed wake effect at the tip section shows that the contribution of the advancing and retreating blade on the force coefficient is smaller than the one of the skewed wake. This can be seen from the fact that the maximum forces are shifted to the upwind side of the blade, which results in a stabilizing yaw moment.

The other possible source of error for this section is the tip correction model. The capability of these correction models in the prediction of the loads in the yawed flow needs to be further investigated.

### 4. Rotor tower interaction

Due to new materials, control strategies and the fact, that noise is offshore not a driving factor, downwind turbines seem to go into a revival. So far Ming Yang, Hitachi and the Dutch developer 2-B Energy have been developing downwind concepts lately [13].

The effect of the tower shadow is a crucial question in the aerodynamics. In BEM based calculations mostly either a wind deficit, from potential theory, or empirical models are used [13, 14]. In this paper it is investigated, how much this assumption holds using CFD simulations.

Again, the NREL Phase VI wind turbine is used for the investigation. For the mesh generation the bladeBlockMesher, developed by ForWind and Fraunhofer IWES was used [8]. It creates a structured blockMesh around wind turbines blades. The rotor was meshed separately in a cylindrical mesh from the surrounding mesh. The latter has been meshed in a half-cylindrical way with the hub distance of 5D to the inlet and crossflow cylinder walls and 10D distance to the outflow. Overall 10 Million cells have been used to discretize the domain. Several grids were
examined in order to minimize the effect of the grid errors on the solution which is discussed in [8].

Fig. 3 shows the numerical domain used in this part. The k-ω SST model has been used for unsteady RANS turbulence modeling [15]. The simulations have been done at an inflow velocity of $U_0 = 7$ m/s with the turbulence intensity of 1%.

![Computed domain with different separated rotor mesh and the overall domain in a half-cylindrical shape.](image)

The tower influence is modeled using the FAST code based on the work of Bak et al. [16], which is based on a potential flow solution around a cylinder. The influence of the tower on the velocity is therefore applied based on the position and the drag coefficient of the tower section.

Fig. 4 shows the simulation results for the normal and tangential force coefficient at the blade sections 30% and 80% for the downwind configuration at $U_0 = 7$ m/s. The CFD simulation shows a good agreement with the experimental data for both the amplitude and the location of velocity drop for both sections. However, the simulation results obtained by FAST shows 20% larger mean values. This deviating result might come from the input-polars. Nevertheless, it should be noted that the simulations using FAST are a standard certified test case provided by NREL, and no modifications were done on that.

![Normal and tangential force coefficient for 30% (upper) and 80% (lower) of span for wind speed of 7 m/s.](image)
Fig. 5 shows a comparison of the lift coefficient at two different sections along the blade span. For extracting the angle of attack (AOA) the technique of Shen [17] is used. This technique for extracting the AoA uses the Biot-Savart integral to determine the impact of the bound vorticity on the velocity field [17]. The approach assumes, that the longitudinal force distribution and the induced velocity caused by numerical calculations at a point of interest near the blade are numerically known. Then, the AOA is computed based on loading and velocity vectors at the point of interest. The results from FAST shows an approximately 20% higher value for the lift coefficient compared to CFD results. The CFD simulations show a drop of 50% compared to the FAST simulation which has a drop of 70%. The speed-up (increase in the force coefficients) which is observed near 170 degree for the experimental data is not captured neither by CFD nor BEM. The reason for that could be due to the fact that time stepping of the simulations were too large to capture this specific aerodynamic effect.

5. Wind turbine blades at standstill
Another complex load case relevant for wind turbine designs is the so called standstill case. This situation can occur during the erection of a wind turbine, during operation due to failure in the control system or very extreme storm cases and can result in very high loads acting on the wind turbine structure.

According to the IEC 61400-1 guideline [18], in this load case the blades have to withstand extreme wind speeds. Due to inflow angles of around 90\(\degree\) and therefore fully separated flows, the standstill scenario is numerically complex to simulate. As the interaction between the incoming wind and the blade structure plays a dominant role in this load case, a fluid-structure coupled solver, which combines the open source CFD code OpenFOAM with an inhouse structural beam solver, is used in this part of the work. Utilizing the Geometrically Exact Beam Theory (GBET) [19] for the structural part and a Delayed-Detached Eddy-Simulation for the fluid part, the coupled solver allows a high fidelity analysis of complex aero-elastic problems.

In this section, the CFD based computations are based on a fully structured mesh generated with the software Pointwise [20]. The domain has the shape of a half sphere and the radius of the latter accounts to seven blade lengths. In total, 24 million cells are being used to discretize the domain in space. Three different meshes were used to check for grid convergence. Fig. 6 illustrates the used computational grid.

Fig. 7 shows a comparison of the developed high fidelity solver with the standard BEM-based engineering tool FAST [21] for the NREL 5MW reference wind turbine [22] during standstill. The inflow angle corresponds to 90\(\degree\), the wind speed is set to 45m/s. Since fully separated flows are also difficult to be accurately captured with RANS, time accurate Delayed-Detached Eddy Simulations (DDES) are performed. The simulations were initialized for 1 second to achieve a realistic field. Afterwards the flow simulation was coupled with the structural solver. In total the simulations were ran for 25 second. It can be noted, that the mean values of both edgewise and
flapwise deflections show a reasonable agreement for both numerical approaches. In contrast to this, a significant difference can be observed in the dynamics of the edgewise deflection. While the FAST result shows an almost constant edgewise deflection, the CFD coupled method predicts clear edgewise vibrations. The reason for this difference can be found in the complexity of the flow at such high angles of attack, which cannot be captured by FAST due to a missing suitable correction model. Fig. 8 gives an impression of the strongly turbulent wake behind the blade.

It can be concluded that FAST could reproduce the mean deflections computed by the coupled CFD approach for the investigated velocity. However, blade vibrations could not be captured by the BEM based approach due to a missing suitable correction model. Potential instabilities, for example caused by overlapping vortex shedding frequencies and natural

Figure 6: Half-cylindrical domain used for the fluid-structure computations of the NREL 5MW blade.

Figure 7: Deflections at blade tip for 45m/s.

Figure 8: Instantaneous isosurface for Q coloured by velocity magnitude. Inflow speed: 45m/s.
frequencies of the structure, could therefore not be captured by the BEM based approach. Following that, results obtained by FAST for this particular case should be treated with care.

6. Conclusions
While BEM can provide sufficient good results in most general load cases, care needs to be taken at cases leading to flow separation. CFD methods can help to get a more complete picture of the aerodynamics even in such complicated cases. Especially simulations of separated 3D flows with CFD give reasonable results. For load cases which are critical and difficult to grasp with BEM, CFD simulations should be used to cross check, if the critical situations are reached. In this work three different cases were simulated. For yawed flow sectional results over a rotor revolution showed a significant difference between different codes. In the root area, the skewed wake model from Schepers coincides more with the results from CFD since this model takes the root vortex into account. Close to the tip the behaviour is closer to the sinusoidal behaviour however some deviation from CFD is still observed, due the effect of advancing and retreating of the blade against the wind direction. The effect of the tower of downwind turbines was rather overestimated by the BEM. The fluid-structure coupled simulations showed that BEM and CFD predict a comparable mean deflection for the NREL 5MW blade during standstill. However, edgewise blade vibrations could be captured by the BEM based code and therefore BEM results for that specific load case should be handled with care.

References
[1] Schepers J 2012 Engineering models in wind energy aerodynamics november ISBN 9789461915078
[2] Micallef D 2012 3D flows near a HAWT rotor : (Technische Universiteit Delft) ISBN 9789995703127
[3] Jonkman J M and Buhl Jr M L 2005 National Renewable Energy Laboratory, Golden, CO, Technical Report No. NREL/EL-500-38230
[4] OpenCFD 2015 Openfoam®- the open source computational fluid dynamics (cfd) toolbox accessed 29 July 2015 URL http://OpenFOAM.com
[5] Schepers J 1998 An engineering model for yawed conditions, developed on the basis of wind tunnel measurements
[6] Hand M M, Simms D, Fingersh L, Jager D, Cotrell J, Schreck S and Larwood S 2001 Unsteady aerodynamics experiment phase vi: wind tunnel test configurations and available data campaigns Tech. rep. National Renewable Energy Laboratory
[7] Spalart P, Deck S, Shur M, Squires K, Strelets M and Travin A 2006 A New Version of Detached-eddy Simulation Resistant to Ambiguous Grid Densities vol 20 (Springer-Verlag)
[8] Rahimi H, Daniele E, Stoevesandt B and Peinke J 2016 Wind Engineering
[9] Patankar S and Spalding D 1972 International Journal of Heat and Mass Transfer 15 1787 – 1806 ISSN 0017-9310 URL http://www.sciencedirect.com/science/article/pii/0017931072900543
[10] Issa R I, Gosman A D and Watkins A P 1986 Journal of Computational Physics 56 66–82
[11] Moriarty P J and Hansen A C 2005 AeroDyn theory manual (Citeseer)
[12] Rahimi H, Hartvelt M, Peinke J and Schepers G J (IOP Publishing)
[13] Fairley P 2003 Two-bladed wind turbines make a comeback accessed 29 March 2016 URL https://www.technologyreview.com/s/528581/two-bladed-wind-turbines-make-a-comeback/
[14] Hagen T R 2011
[15] Menter F and Esch T 2001 16th Brazilian Congress of Mechanical Engineering pp 26–30
[16] Bak C, Aagaard Madsen H and Johansen J 2001 Influence from blade-tower interaction on fatigue loads and dynamics (poster) (WIP Renewable Energies) pp 394–397 ISBN 88-900442-9-2
[17] 2006 determination of angle of attack (AOA) for rotating blades peinke, Schaumann, Barth (eds). Springer: Oldenburg, Germany,
[18] IEC 2005 Wind Turbines - Part 1: Design Requirements, IEC 61400-1 (3rd edn.) (Genova) ISBN 2-8318-8161-7
[19] Cardona A and Géradin M 1988 International Journal For Numerical Methods in Engineering 26 2403–2438 ISSN 0029-5981
[20] 2015 Pointwise URL www.pointwise.com
[21] Jonkman J 2013 51st AIAA Aerospace Sciences Meeting, Grapevine Texas January ISBN 9781624101816
[22] Jonkman J, Butterfield S, Musial W and Scott G 2009 Definition of a 5-MW reference wind turbine for offshore system development Tech. rep.