Numerical simulations of a large offshore wind turbine exposed to turbulent inflow conditions

Galih Bangga, Giorgia Guma, Thorsten Lutz and Ewald Krämer
Institute of Aerodynamics and Gas Dynamics (IAG), University of Stuttgart, Germany
bangga@iag.uni-stuttgart.de

Abstract – The present works are intended to investigate the aerodynamic responses of a large generic 10MW offshore wind turbine under turbulent inflow conditions. The non-linear Lifting Line Free Vortex Wake Simulations approach is employed for this purpose computed using the QBlade code. In these studies, the effects of a three-dimensional (3D) correction model for the airfoil polars were studied in advance. For this purpose, the Blade Element Momentum computations employing the corrected polars are performed and compared to Computational Fluid Dynamics (CFD) simulations, and a good agreement is obtained between both employed approaches. Background turbulence is then imposed in the QLLT simulations with the turbulence intensities ranging from low to high turbulence levels (3% - 15%). Furthermore, the impact of wind shear from different locations (offshore and onshore) is investigated in the present works.

1. Introduction

A fundamental issue in accurate estimation of the power output has been noted with the continuous increase of the offshore wind farm size which is partly contributed by difficulties in flow and wake modeling [1]. This is particularly caused by the complexity of the wake downstream of the turbine and their relationship with atmospheric variables such as the variability of wind speed, direction, turbulence and atmospheric stability that is not yet fully understood [2]. Further understanding of the relationships between these variables is required to improve the current state of the art wind farm and wake models.

Several studies regarding wind turbine wakes have been carried out for both offshore and onshore applications. It has been noted that the atmospheric variables can have a strong influence on the wake recovery distance and the mixing process of the turbulent flow. Barthelmie et al. [3] presented numerical simulations and measurements of power losses and turbulence intensity in wind turbine wakes at an offshore wind farm. They showed that the wake model could deliver reasonable predictions on these parameters as compared to the experimental data. Furthermore, they noted that wake loss increases with decreasing wind speed. The turbulence level was shown to increase with wind speed. Mycek et al. [4] and Kim et al [5] pointed out the ambient turbulence intensity rate has a minor influence on the mean performances of a marine current turbine. However, the performance fluctuations dramatically increase with the turbulence intensity and may have a major impact on the fatigue loads on the machine, and consequently on its maintenance and manufacturing costs. Furthermore, it was shown in [5] that the ambient turbulence enhances the mixing process in the wake reducing its overall recovery distance from the turbine.

When regarding wakes, a distinct division can be made into near and far wake regions [6]. The first mentioned category is close to the rotor where the main characteristics of the rotor properties can be observed. In the far wake area, the mean wind profile becomes Gaussian and the influence of surrounding environment is more dominant. A simple expression for this division was made by
Sørensen et al. [7] and the same approach was applied by Kim et al. [5] using CFD approaches. The latter was carried out on the generic 10MW AVATAR turbine [8] where the rotor diameter reaches 205.8 m, and the same turbine is studied in the present investigations.

The main aim of the present works is to further study the wake characteristics of the large AVATAR rotor under turbulent inflow conditions employing the QBlade code. In contrast to the works made by Kim et al. [5] that used computationally expensive CFD simulations, the present works employ a simplified approach using the non-linear Lifting Line Free Vortex Wake Simulations. This method is denoted as QLLT in the QBlade code. The accuracy of the method in modeling the underlying complex flow behavior in the wake is investigated by comparing the results to full scale CFD computations. The turbulence levels range from 3% to 15%. Three dimensional effects in the root area of the rotor are included in the simulations as these are important especially for the flow characteristics in near wake area [6,9,10]. The accuracy of the employed 3D stall delay model is studied in advance by comparing Blade Element Momentum (BEM) simulations to fully resolved high fidelity CFD simulations in 120° simulation model.

![Figure 1: Off-shore wind turbine specification for the AVATAR rotor geometry.](image)

2. The AVATAR Rotor and Operating Conditions

The generic 10MW AVATAR blade [8] was chosen in the present examination. The blade is twisted and tapered. The rotor was designed as a variation of the DTU 10MW reference turbine with a lower induction than the DTU 10MW reference turbine. A scaling factor of 1.15 was applied in the DTU 10MW rotor resulting in the rotor diameter of 205.8 m. An illustration of the studied wind turbine is given in Fig. 1. The rotor operates at a wind speed of 10.5 m/s which is defined as the standard test case for the AVATAR rotor and at a rotational speed of 9.02 rpm. Four different inflow turbulence levels (3%, 6%, 10% and 15%) as well as a case without turbulence were considered in the present studies. Furthermore, the impact of wind shear from different topographies (maritime, flat terrain and agricultural-land) combined with inflow turbulence on the rotor performance is considered. Four different logarithmic wind shear profiles with the surface roughness height \(z_0\) ranging from 0.0001 m to 0.1 m are simulated. The shear was generated based on the wind speed of 10.5 m/s measured at the hub height of 132.7 m.

3. BEM and CFD Simulations

Before the background turbulence was imposed in the QLLT computations, additional BEM computations were performed and compared to CFD simulations in 120° (1/3) simulation model to
assess the accuracy of the employed stall delay model on the load predictions. The BEM computations were also carried out using the open-source QBlade code [11] developed at the Technical University of Berlin, Germany while the CFD simulations were carried out using the FLOWer code [12] from the German Aerospace Center. The FLOWer code has been extended at the IAG during the last years for wind turbine applications. Turbulence closure of the URANS equations in the present studies is provided by the eddy viscosity two equation Shear Stress Transport (SST) $k-\omega$ model since the model is able to give good predictions for flows experiencing an adverse pressure gradient. Fig. 2 presents the mesh used for the 3D CFD computations. The mesh of the blade is of C-H type and was constructed using the commercial grid generator Gridgen with the “automesh” script developed at the institute. The blade mesh consists of 281x129x193 grid points in chordwise, normal and radial directions, respectively. The first grid cell was set to meet $y^+ < 1$ to resolve the laminar sub layer zone; and 32 cells are located across the boundary layer. Further details of the 3D mesh and the CFD setup are given in [9].

For the BEM computations, a 3D correction model according to Chaviropolous and Hansen [13] was considered, and the results for the sectional tangential force ($F_T$) and power curve are presented in Fig. 3. It can be seen that the aerodynamic force and power predictions are improved by the inclusion of the stall delay model while the pure BEM calculations significantly underpredict the results especially for $F_T$ in the inboard area and at high wind speed cases. The same behavior was documented in [9]. On the other hand, it was observed that drag is not severely influenced by the correction but its impact is considered small compared to the overall effects of the lift force.

![Figure 2: Fully resolved blade mesh used in the present CFD calculations.](image1)

![Figure 3: Predicted sectional tangential force (left) and power curve (right) of the rotor. The CFD result for the power curve (right figure) is obtained from [8].](image2)
Figure 4: Power fluctuations for various turbulence levels (left) and comparison to CFD data in terms of the power spectra for the case with TI = 10%. The raw data of the CFD results are obtained from [17].

Figure 5: Wake structures visualized by $\lambda_2$-criterion of $-0.01 \text{ s}^2$. It can be seen that stronger wake fluctuations and earlier breakdown of the wake vortices occur for the rotor exposed the higher turbulence intensity.

4. Effects of Background Turbulence

In this section, the QLLT simulations were carried out for the AVATAR rotor exposed to four different background turbulence levels which were generated using the Veers method [15]. The wind field was generated and interpolated in between the spatial grid and temporal points, and it was marched during every time-step of the simulation where frozen turbulence was assumed for this treatment [16]. The simulations were carried out for 200 s (30 rotor revolutions) using the time-step size of 0.369 s. It is worth mentioning that the tower and ground were not modeled in the studies to isolate the inflow turbulence effects.
Power fluctuations for various turbulence levels are plotted in Fig. 4 (left). It can be seen that stronger fluctuations are observed for the rotor exposed to a higher turbulence level. The peak amplitude of the rotor power can even reach as high as 5 MW for the turbulence level of 15%. In Fig. 4 (right), the results of the QLLT simulations are compared to available CFD results from [17], where time series data of the rotor power are available. The CFD computations were computed at IAG using the FLOWer code as a part of the AVATAR project task. It shall be noted that the turbulent inflow in CFD was generated using the Mann box approach [18] in contrast to the QLLT simulations where the Veers method [15] is used. The turbulent wind field was constructed in the free vortex simulations using three different spatial resolutions (Δ): 40 m, 20 m and 13.3 m. It is worth noting that the results presented in Fig. 4 (left) and for the rest of the paper uses Δ = 13.3 m. As can be seen, both QBlade and CFD have a very good agreement in terms of the predicted power spectra regardless of Δ especially up to frequency of 1 Hz, but the QBlade results for the higher frequency seems a bit non-physical as no decay is observed. The dominant peaks shown by the CFD data in Fig. 4 (right) are clearly reproducible by the QLLT simulations even though the amplitude drop of the low frequency spectra is overestimated if Δ is too large, see black and purple boxes in Fig. 4 (right). Nevertheless, the main characteristics of the rotor power influenced by external turbulence are well captured.

In Fig. 5, the wake structures downstream of the rotor are presented. It can be seen clearly that earlier breakdown of the tip vortices occurs for the case with a higher turbulence level, indicating the importance of the external perturbations on the turbulent mixing in the wake area. The wake of the higher turbulence level case is also characterized by the strong vortex fluctuations that may alternate the inflow wind speed and direction for the downstream turbine. A careful treatment regarding this issue needs to be considered since the resulting fluctuations may cause a greater vibration than the upstream rotor endangering the turbine structure itself. Despite that, the higher turbulence level speeds up the recovery of the wake as presented in Fig. 6. This renders a conclusion that the downstream turbine can achieve a higher power production if the upstream turbulence is greater. The trade-off between the power production and load fluctuations may be reduced by applying an appropriate active control system.
5. Effects of Wind Shear

In this sections, the impact of surface roughness from different topographies (maritime, flat terrain and agricultural-land) combined with inflow turbulence on the rotor performance is investigated. Four different logarithmic wind shear profiles as already mentioned in Section 2 are considered. The aforementioned roughness height corresponds to the different topography. For example, $z_0 = 0.0001 \text{ m}$ represents a calm water surface or within the offshore environment and $z_0 = 0.1 \text{ m}$ is for an agricultural land with few buildings. For these studies, the turbulence intensity of 10% is applied.

Fig. 7 presents the power and thrust fluctuations over the time and its corresponding frequency spectra for various shear profiles. No definite effects of the shear profile on power and thrust fluctuations can be observed in Fig. 7 (upper). This statement is supported as well by the frequency spectra in Fig. 7 (lower). The reason is expected to be related to the applied turbulence level which seems to have a stronger contribution to the wind turbine loads than the logarithmic wind shear. Despite that, the logarithmic shear seems to have a quite prominent influence in the wake area. It can be seen in Fig. 8 that the streamwise velocity magnitude near the ground reduces with increasing roughness height. Again, similar to the high turbulence intensity effects, the downstream turbine can be affected by this velocity deficit. The expected power increase for the high turbulence level may be reduced.

![Diagram showing power and thrust fluctuations over time and frequency spectra for various wind shear profiles.](image)

Figure 7: Power and thrust fluctuations for various wind shear profiles.
6. Conclusions

Numerical simulations have been carried out to assess the performance of a large offshore wind turbine under turbulent inflow conditions employing non-linear lifting line free vortex approaches. This simplified model was used to calculate the rotor power, thrust and wake characteristics of the generic 10MW AVATAR turbine. In these works, additional fully resolved 3D CFD simulations were also performed to evaluate the accuracy of the employed 3D correction model, where a very good agreement is obtained.

The free vortex simulations for the turbulent inflow case were carried out for four different turbulence intensities ranging from 3% up to 15%. It was been shown that the 3D CFD result for power fluctuation is accurately reproduced by the simplified model for the employed spatial resolution in the inflow turbulence generation. The higher turbulence level initiates an earlier breakdown of tip vortices, facilitating a faster wake recovery that may benefit the downstream turbine. Despite that, a stronger fluctuation of the wake velocity and direction was also observed. The trade-off between these variables needs to be considered in a wind farm analysis.
The present studies reveal that wind shear has little influence on the global aerodynamic loads acting on the turbine. It seems that the relatively high turbulence level imposed at the inlet has a stronger influence than the wind shear. Despite that, it is evident that wind shear affects the velocity field in the wake area, increasing the velocity deficit especially near the ground. This may reduce the tendency of the power increase potential for the downstream turbine at a high turbulence level.

Acknowledgement
The AVATAR project consortium is gratefully acknowledged for provision of the blade geometry and data.

References