Studying the effect of blade deflections on the aerodynamic performance of wind turbine blades using OpenFOAM

B. Dose¹, B. Stoevesandt², J. Peinke¹,²
¹ ForWind - University of Oldenburg, Institute of Physics, Oldenburg, Germany, bastian.dose@forwind.de; ² Fraunhofer IWES, Oldenburg, Germany;

Summary

Modern wind turbine rotor blades are designed increasingly large and flexible. The increasing flexibility can lead to problems in the field of Computational Fluid Dynamics (CFD), as the geometry deformations cannot be captured. In the current work we developed a new, coupled framework to investigate flexible rotor blades. As an example, simulations are performed on the NREL 5 MW reference turbine [4] to study the effect of the blade deflections on the aerodynamic performance of wind turbine blades.

1. Introduction

Modern wind turbine rotor blades are designed increasingly large and flexible. This structural flexibility represents a problem for the field of Computational Fluid Dynamics (CFD), which is used for accurate load calculations and detailed investigations of rotor aerodynamics. As the blade geometries within CFD simulations are considered stiff, the effect of blade deflections caused by aerodynamic loads cannot be captured by the common CFD approach. Coupling the flow solver with a structural solver can overcome this restriction and enables the investigation of flexible wind turbine blades by means of CFD.

2. Methodology

To capture the effect of wind turbine blade deformations within CFD simulations a new Finite Element (FE) solver was implemented into the open source CFD code OpenFOAM [1]. To obtain a compromise between accuracy and solver performance for the structural solver, non-linear finite beam elements are used to account for the blade structure. The implementation of the latter is based on the Geometrically Exact Beam Theory (GEBT), originally proposed by Simo [2], and the work of Li [3]. While the use of 1D-beam elements limits the number of structural degrees of freedom and therefore the computational effort required for solving, the geometrical non-linear formulation of the element ensures even for large deformations and rotations of the blade structure a sufficient accuracy. In addition, the implemented formulation supports material anisotropy.

To couple both flow and structural solver, a partitioned coupling approach is implemented. This ensures the flexibility of the developed framework, as different existing OpenFOAM flow solvers can be coupled to the structural solving module. The implemented steady-state coupling procedure is illustrated in Figure 1.

![Figure 1: Coupling procedure](image-url)
The solving of the flow field and the mesh update run fully parallelized, whereas the response of the structure is solved in serial as it takes only few seconds. Because flow and structural solvers are implemented within one combined C++ framework, information transfer is performed directly in memory and without the need of an external coupling interface. Combined with structural cross section analysis tools, the new framework represents a powerful tool for aerodynamic investigations of flexible wind turbine blades.

3. Setup

To test the developed solver, the NREL 5 MW baseline wind turbine [4] is simulated. It was designed as a conventional horizontal axis wind turbine and has a rotor diameter of 120 m. To improve the given reference geometry, the description is slightly modified based on the thesis of Martin [5].

To simplify the simulation setup, the cone and tilt angles are removed from the reference design and the rotor of the wind turbine is simulated without the tower and nacelle. Because a uniform inflow conditions is used, the symmetry of the rotor can be utilized to reduce the number of cells in the flow domain. The dimensions of the computational domain (in units of rotor diameters) are illustrated in Figure 2.

The meshing of the domain is generated using the meshing software Pointwise [6]. Using a hybrid mesh approach, structured mesh regions are combined with unstructured parts to obtain a compromise between effort needed for the generation of the mesh and grid quality. The separated mesh regions are connected by grid interpolation interfaces.

To ensure a high resolution of the calculated flow field, in total 18 million cells are used for the domain. The boundary layer near the blade surface is resolved with nine million cells; the near wake and farfield are resolved with six and two million cells respectively. The normalized wall distance accounts to \( y^+ \approx 100 \) for the blade surface. As an example, the structure of the computational grid around the mid-section of the blade is shown in Figure 3.

![Figure 3: Used mesh around blade surface](image_url)

To obtain the solution for the flow field, the incompressible Reynolds-Averaged Navier-Stokes (RANS) equations are solved using the two equation k-\( \omega \) SST turbulence model and adaptive wall functions. The simulation of the rotor within the steady-state simulation is realized by using a rotating reference frame. For the structural domain, 48 two-node non-linear GEBT beam elements are used. The sections properties are obtained from the official NREL 5 MW reference report and gravitational forces are neglected.

4. Results & Discussion

In order to verify the developed framework and to investigate the effect of blade deformations, two different setups are simulated and compared to the NREL reference report. For case 1, the rigid rotor blade is investigated, for case 2 the
deformation of the blade is taken into account. Both cases are simulated for rated conditions, which are described by a zero pitch, uniform inflow velocity of 11.4 m/s and a rotational speed of 12.1 rpm. Case 1 is performed using the steady-state solver ‘simpleFOAM’ of the OpenFOAM toolbox, case 2 is simulated using the new, fluid-structure coupled framework.

Table 1 lists the calculated values from the simulations and the reference values from the NREL report. It should be noted that the NREL reference results are based on lower fidelity models, as no experimental data exists.

<table>
<thead>
<tr>
<th></th>
<th>NREL report</th>
<th>CFD flexible</th>
<th>CFD rigid</th>
</tr>
</thead>
<tbody>
<tr>
<td>Power [MW]</td>
<td>5.30</td>
<td>5.60</td>
<td>5.54</td>
</tr>
<tr>
<td>Thrust [kN]</td>
<td>790</td>
<td>778</td>
<td>749</td>
</tr>
</tbody>
</table>

**Table 1: Aerodynamic operating numbers**

Compared to the provided reference results, the simulation results obtained from the new, coupled solver show reasonable agreement. Although power is slightly over predicted by 5.6 %, the rotor thrust force is in the same range as the reference result. In addition, both flapwise and edgewise tip displacements of the blade structure are in the same range as in the reference (listed in Table 2). Similar results were also reported by Imliea et al. [7].

<table>
<thead>
<tr>
<th></th>
<th>NREL report</th>
<th>CFD flexible</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flapwise</td>
<td></td>
<td></td>
</tr>
<tr>
<td>displacement</td>
<td>5.40 m</td>
<td>5.89 m</td>
</tr>
<tr>
<td>Edgewise</td>
<td>0.61 m</td>
<td>0.63 m</td>
</tr>
</tbody>
</table>

**Table 2: Structural deformations of the blade**

The wake structure behind the rotor is illustrated in Figure 4, where the characteristic tip and rot vortices can be observed.

![Figure 4: Rotor wake; Isosurface for $\lambda_2$, coloured by the velocity magnitude](image)

Figure 5 shows the shape of the deformed blade for rated conditions more in detail.

![Figure 5: Deflected and initial (in grey) rotor blade geometry; Isosurface for $\lambda_2$, coloured by the velocity magnitude](image)

The predicted large flapwise deformation of the outer blade region is clearly visible.

When comparing the results of the rigid and flexible CFD simulation, as listed in table 1, it is counter intuitive that both power and thrust are increased for the flexible blade. As the rotor diameter decreases by approximately 0.85 m due to the blade deformation, lower values could
be expected. The reason for this can be found in the torsion of the outer part of the blade.

Reaching its maximum at the blade tip with a torsional deformation of 1.3 °, the increased angles of attack result in larger forces. Figure 6 shows the angles of attack in the outer blade region for both simulated cases. The extraction of the angles from the CFD results is performed using the azimuthal averaged technique [8].

![Figure 6: Extracted Angle of Attack](image)

5. Conclusion

A new framework for steady-state CFD investigations of flexible wind turbine blades has been presented. The coupled tool allows efficient CFD simulations of large and flexible wind turbine blades. As an example, a fluid-structure coupled analysis of the NREL 5 MW baseline turbine has been shown. Compared to the reference result, a reasonable agreement was found using the new developed solver. The predicted increased power and thrust production of the deformed blade geometry can be explained by the increased angle of attack due to the torsional deformation of the outer blade structure.

Acknowledgement

The present work is funded within the framework of the joint project Smart Blades (0325601D) by the German Federal Ministry for Economic Affairs and Energy (BMWi) under decision of the German Federal Parliament. We would like to thank the computer time provided by the Facility for Large-scale computations in Wind Energy Research (FLOW) at the University of Oldenburg.

In addition, the authors would like to thank Mike McWilliam from the University of Victoria, Canada for the help with the GEBT theory and Manfred Imiela from DLR Braunschweig for the help with the geometry of the NREL 5 MW reference wind turbine.

References