

Reply to comments by Reviewer 1

Pascal Weihing on behalf of the authors
IAG, University of Stuttgart

July 17, 2018

The authors would like to thank Prof. Smaili for his efforts and valuable comments. They are very much appreciated and incorporated into the revised paper.

In the present document the comments given by the 1st reviewer are addressed consecutively. The following formatting is chosen:

- The reviewer comments are marked in blue and italic.
- The reply by the authors is in black color
- A marked-up manuscript is added. Changed section with regard to the comments by reviewer 1 are marked in yellow. Changed sections with regard to comments by both reviewers are marked in green. Highlighting in gray denotes passages that have been changed by the authors in order to improve the clarity or the argumentation but which are not related to specific reviewer comments.

General comments

1. *"The flow in the nacelle region of a wind turbine has been the subject of several previous studies. Thereby, the authors may improve the literature review, a more serious bibliographical search and study might be carried out."*

A new subsection has been added in the introduction which addresses the interacting flow fields of the rotor and the nacelle **R1:G1** (page 3, line 66). The following references have been added:

Masson, C., & Smaili, A. (2006). Numerical study of turbulent flow around a wind turbine nacelle. *Wind Energy: An International Journal for Progress and Applications in Wind Power Conversion Technology*, 9(3), 281-298.

Zahle, F., & Sørensen, N. N. (2011). Characterization of the unsteady flow in the nacelle region of a modern wind turbine. *Wind Energy*, 14(2), 271-283.

Johansen, J., Madsen, H. A., Sørensen, N. N., & Bak, C. (2006). Numerical Investigation of a Wind Turbine Rotor with an aerodynamically redesigned hub-region. In 2006 European wind energy conference and exhibition, Athens, Greece.

2. *"The mesh study was not presented: the choice of grid type and size was not justified"*

The authors agree that an assessment of the accuracy of numerical predictions is very important, particularly if there is no reference data available to validate the results. The grids in the present

study are based on experiences gained during many national (AssiST, DFG-PAK780, LARS, TremAc, OWEALoads) and international research projects (MexNext, AVATAR, Innwind) and are based on the recommendations for the cell spacings and growth rates made during the NASA drag prediction work shops. In order to check for the influence of the grid on the solution in the present study a very fine grid has been employed for comparison which is planned to be used for future DES simulation. The trend on the sectional load distribution shows that there is only a very small grid influence. The sectional thrust curves more or less collapse completely. For the sectional driving force, very small deviations are visible in the radial distribution. Interestingly, the differences in the integral driving force is one order of magnitude smaller compared to the differences in the integral thrust. For this reason a classical grid convergence study (GCI) can be sometimes misleading, since local effects might be caught up by error compensation. However, these local effects are particularly important the present case where a detailed analyses of three-dimensional features are studied. Probably, a "bad" grid in the root region with for example large skew angles, aspect ratios or under-resolved boundary layers would not allow for a detailed evaluation of the relevant flow features, but on the other hand would also not be reflected in a global GCI.

Although, the general impact of the grid on the solution seems to be very small, it must be noted that for the lower wind speeds the local effects of the aerodynamic modifications on the overall blade performance can come into the same order of magnitude as the accuracy of the CFD framework. This fact is analyzed in the newly introduced section 4.6.

Regarding the grid dependency analysis, the paper has been modified in **R1:G2-a** (page 7, line 181) and **R1:G2-b** (page 10, line 224).

3. *"The use of a compressible Navier-Stokes solver should be justified. It would be desirable to present the contours of the Mach number."*

It is certainly clear that the use of a compressible flow solver is not necessary when dealing with flow features in the very inboard region of a wind turbine. In the present cases the maximum Mach numbers on the suction side of the airfoils in the tip region was around 0.29. The simple reason, why we use this solver is that it is the only available one within the code FLOWer. FLOWer on the other hand is a well proven CFD code that has been applied in numerous aerodynamic and aeroelastic studies of wind and helicopter applications, so that a lot of experience has been gained. The code is continuously further developed at the authors institute. A list of references can be found at

https://www.iag.uni-stuttgart.de/abteilungen/luftfahrzeugaerodynamik/veroeffentl_luftf/veroeffentlichungen_luftf.index.html

An additional fact to add is that in the future, compressibility effects will play a more important role, when the turbine diameters increase or when for example offshore higher tip-speeds might be realized. For the high tip speed cases of for example the DTU 10MW rotor, or even for the small scale MEXICO turbine over-predictions of the lift in the outer portion of the rotor at higher angles of attack predicted by some incompressible codes could be explained by the neglect of compressibility. When taking into account compressibility the adverse pressure gradient increases which leads to an earlier separation compared to an incompressible assumption. An investigation on that has been a task of the AVATAR project, where a comparison of FLOWer and the EllipSys code focusing on the effects of compressibility and possible corrections has been conducted and which was recently presented within a collaborative paper.

Sørensen, N. N., Bertagnolio, F., Jost, E., & Lutz, T. (2018, June). Aerodynamic effects of compressibility for wind turbines at high tip speeds. In *Journal of Physics: Conference Series* (Vol. 1037, No. 2, p. 022003). IOP Publishing.

4. *"The validity of the numerical simulations was not presented."*

This is correct, since no measurement data was available for validation. Additional references have been placed in section 3.1 **R1:G4** (page 5, line 144), which state that the present numerical methodology has given accurate results in other projects, where measurements were available and code-to-code comparisons have been performed.

5. *"The full-turbulence models are not suitable for describing the flow fields in the hub and nacelle region; because probably in such situation, boundary layer transition may occur, and therefore conclusions drawn might be far from reality. "*

The authors agree that boundary layer transition might affect the development of flow separation in the hub region. Accounting for the laminar flow history in the front part of the blade will probably lead to a downstream shift of the separation, as the shear stress of the “freshly” transitioned boundary layer is higher compared to the boundary layer which was turbulent right from the beginning. In order to evaluate this hypothesis and to check whether the entire flow pattern changes, or not, transitional simulations have been performed for the baseline geometry. Two transition models have been chosen to draw a comparison with the fully turbulent results: The correlation based $\gamma-Re_\theta$ model as well as the e^N envelope model, both coupled to SST closure. The results are presented in Appendix B. As expected, flow separation diminishes by including the effect of transition. However, the main flow topology stays the same. For the production runs comparing the different geometries, boundary layer transition was omitted for the following reasons:

- Reduction of additional model uncertainty: The effects induced by the geometrical modifications on the overall rotor performance is rather small. Additional uncertainties stemming for example from slight deviations in the transition location (which might be unsteady) are unfavorable.
- There is no engineering transition model that can accurately account for the mechanisms describing boundary layer transition in corner flows. In that region strong cross flow prevails which neither the state-of-the art $\gamma-Re_\theta$ model nor the e^N envelope method accounts for.
- Experimental studies such as those of

Zamir, M. "Similarity and stability of the laminar boundary layer in a streamwise corner." Proc. R. Soc. Lond. A 377.1770 (1981): 269-288

suggest that boundary layer transition in corner flows occurs earlier than in equivalent conditions over a flat plate.

- In reality, pollution and erosion leads to an earlier transition than predicted by the standard models that do not include roughness effects.

Hence, with respect to the latter two points the actual transition location and the flow field development can be expected to be somewhat in between the fully turbulent simulations and the transitional cases presented in Appendix B. These deliberations can be found in the revised paper in **R1:G5-a** (page 5, line 154), whereas the results comparing fully turbulent and transition simulations are presented in **R1:G5-b** (page 38, line 711)

6. *"To obtain more relevant conclusions, the simulations should be carried out for other wind speed values, not only for 10 m/s."*

Thank you for pointing that out. The additional wind speeds 8, 12 and 15m/s have been analyzed for the baseline and the optimized geometry. In particular for the higher wind speeds

valuable information on the stall mechanisms and on the global load behavior could be deduced. These additional cases can be found in the newly introduced section 4.6. The relevant text passages added are **R1:G6-a** (page 9, line 216), **R1:G6-b** (page 33, line 602) and **R1:G6-c** (page 38, line 684).