

This document includes:

1. Point-to-point response to the first reviewer
2. Point-to-point response to the second reviewer
3. List of the major changes in the manuscript
4. Marked-up manuscript
 - changed sections with regard to the comments by reviewer 1 are marked in yellow
 - changed sections with regard to the comments by reviewer 2 are marked in cyan
 - changed sections with regard to the comments by both reviewers are marked in green
 - changes with regard to no comments but which serve a better understanding and an improvement of the manuscript are marked in gray

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).

Reply to comments by Reviewer 2

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

July 17, 2018

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

In the present document the comments given by the 2nd 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 2 are marked in cyan. 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. *"In the introduction, it would be useful to mention other techniques that can be used to improve aerodynamic efficiency in the root region with flatback airfoil, such as flow control devices (splitter, cavity, flap...), and to show how the authors' solutions can stand out. "*

This point has been added to the introduction in section **R2:G1-a** (page 3, line 81) by mentioning measures to increase the efficiency of conventional root sections and root sections with flatback airfoils. For the cylinder like sections active and passive flow control technologies such as VGs, or blowing/plasma actuators might be utilized in future applications. For flatback airfoil sections the causes of the base drag are briefly described and possible solutions are listed such as Gourney flaps, splitter plates, or cavities. The relevant literature describing all these measures is given. In addition, a clearer distinction of the scope of the present work from these efficiency boosting technologies is given in section 1.5. **R2:G1-b** (page 4, line 97)

2. *"With no validation of the model and without a grid independency study, how can you be sure about the results accuracy?"*

This point was also argued by Reviewer#1, so that a critical examination of this has been conducted. 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 **R2:G2-a** (page 7, line 181) and **R2:G2-b** (page 10, line 224).

3. *"Are there any manufacturing constraints for the proposed solutions to increase aerodynamic efficiency?"*

This study is based on results of the research project AssiSt which was a cooperation with industry. The purpose was to get a better understanding of the basic nacelle parameters and the root aerodynamics of the root region of a wind turbine being equipped with flatback airfoils. At first instance the study neglected any other engineering disciplines. Therefore, no strict geometric constraints were imposed. Some considerations, like the lateral blade shifting, would impose a fundamental re-engineering of the engine construction.

Other modifications, primarily the blade root fairing, would easily be applicable from an engineering point of view since the blade connectors are fixed to the spinner. However, this leaves aside any economic deliberation.

An overview on the project results can be found in

Kühn, Timo, et al. "Results of the research project AssiSt." Journal of Physics: Conference Series. Vol. 1037. No. 2. IOP Publishing, 2018.

4. *"The proposed nacelle modifications and their impacts locally, on the root flow have been shown in detail. However, it would be interesting to quantify the impact of these solutions on a global parameter performance, such as the total power produced of the turbine"*

This comment is off course very important, since the crucial parameter that counts is the amount of energy extracted from the wind! Whenever it was possible, the relative improvements or degradations of the rotor efficiency have been added in the text for each of the analyzed modifications:

- regarding the difference of the baseline geometry to the isolated rotor **R2:G4-a** (page 18, line 385)
- regarding the impact of the nacelle thickness **R2:G4-b** (page 22, line 454)
- regarding the relative movement of the blade relative to the nacelle **R2:G4-c** (page 28, line 540)

- In section 4.6 the global assessment of the fairing type modification has been analyzed. Additional wind speeds have been considered that mimic off-design conditions. While the benefits obtained at low wind speeds are very small, they are considerable at higher wind speeds, where additionally a favorable effect can be expected for future investigations that take into account high levels of atmospheric turbulence. **R2:G4-d** (page 33, line 602) and **R2:G4-e** (page 38, line 684)

List of the major changes in the manuscript

The line numbers correspond to the marked-up manuscript, not of the revised version of the manuscript.

- Affiliation: Removed Typo in the address of the authors Kühn and Altmikus
- Abstract: small changes in the wording
- Introduction:
 - Added the subsection 1.3 (page3, line 66) focusing on literature dealing with the interacting flow fields of the rotor and the nacelle
 - Added the subsection 1.4 (page3, line 81) focusing on literature dealing with possible technologies that can be utilized to improve the aerodynamic efficiency in the root region.
 - In subsection 1.5 (page4, line 97) a clearer distinction of the scope of work is made with respect to section 1.4
- In section 2 a more detailed description of the reference turbine is included (page4, line 109)
- Computational details
 - Further references are given that shall prove the validity of the employed CFD solver and framework. (page 5 line 144)
 - The effect of boundary layer transition on the aerodynamics of the root region is discussed (page 5 line 154 – page 6 line 167)
- Results
 - The influence of the numerical grid on the solution is presented in the new subsection 4.1 (page 10, line 224-236).
 - The effect of the geometrical modifications on the overall performance has been added and edited in (page 18, line 385-389), (page 22, line 454-460), (page 28, line 540-548), (page 34, line 625-644)
 - An assessment of other wind speeds (8, 12, 15m/s) apart from the design wind speed is included in section 4.6 (page 33 from line 601)
- Conclusions
 - A summary of the results in off-design conditions as well as a characterization of the overall benefit of the considered modifications is included (page 38, line 684-692)
- Appendix B
 - A comparison between fully turbulent and transitional simulations is presented (page 38 line 711-739).

Numerical Analyses and Optimizations on the Flow in the Nacelle Region of a Wind Turbine

Pascal Weihing¹, Tim Wegmann¹, Thorsten Lutz¹, Ewald Krämer¹, Timo Kühn², and Andree Altmikus²

¹Institute of Aerodynamics and Gas Dynamics, University of Stuttgart, Pfaffenwaldring 21, 70563 Stuttgart, Germany

²Wobben Research & Development GmbH, Borsigstraße 26, 26607 Aurich, Germany

Correspondence: Pascal Weihing (weihing@iag.uni-stuttgart.de)

1 **Abstract.** The present study investigates the flow dynamics in the hub region of a wind turbine focusing on the influence
2 of the nacelle geometry on the root aerodynamics by means of Reynolds averaged Navier-Stokes simulations with the code
3 FLOWer. The turbine considered is a generic version of the Enercon E44 converter incorporating blades with flatback-profiled
4 root sections. First, a comparison is drawn between an isolated rotor assumption and a setup including the baseline nacelle
5 geometry, in order to elaborate the basic flow features of the blade root. It was found that the nacelle reduces the trailed
6 circulation of the root vortices and improves aerodynamic the efficiency for the inner portion of the rotor, but on the other
7 hand induces a complex vortex system at the juncture to the blade that causes flow separation. The origin of these effects
8 is analyzed in detail. In a second step, the effects of basic geometric parameters describing the nacelle have been analyzed
9 with the purpose to increase the aerodynamic efficiency in the root region. Therefore, three modification categories have been
10 addressed, where the first alters the nacelle diameter, the second varies the blade position relative to the nacelle and the third
11 comprises modifications in the vicinity of the blade-nacelle junction. The impact of the geometrical modifications on the local
12 flow physics are discussed and assessed with respect to aerodynamic performance in the blade root region. It was found that
13 increasing the nacelle diameter deteriorates the root aerodynamics, since the flow separation gets more pronounced. Possible
14 solutions identified to reduce the flow separation are a shift of the blade in direction of the rotation or the installation of a
15 fairing fillet in the junction between the blade and the nacelle.

16 1 Introduction

17 In recent years, upscaling of wind turbines has led to steadily increasing rotor diameters. Concepts beyond 10MW reveal
18 diameters of more than 200m (Bak et al., 2017). With that development and the particular fact that aero-acoustic emissions
19 and compressibility effects constrain the tip speed to around 80m/s, a possible measure to increase the overall rotor power
20 could be an increase of the aerodynamic forces in the inner sections of the rotor. However, structural demands and geometric
21 compatibility are dominant design factors over aerodynamic efficiency, so that typically airfoils with high relative thickness
22 are employed which are blended to cylindrical cross sections towards the hub. The poor aerodynamic performance of these
23 sections involving massive flow separation and complex three dimensional flow structures are well known. It can be shown
24 that by assuming a linear increase of the axial induction towards its optimum value along the radius for the inner one third

25 of the rotor, C_P cannot exceed a value of $43/81$. This means a loss of 10.5% compared to the Betz value. In practice, the
26 annual energy production losses of conventional turbines due to flow separation in the root region can be estimated to around
27 3.5% (Loganathan and Gopinath, 2018). In order to increase the rotor efficiency in its inner portion, by an adapted aerodynamic
28 design, a better understanding of underlying flow physics, in particular of the driving effects for flow separation is necessary.

29 **1.1 Flow Separation on Conventional Blade Root Geometries and Rotational Effects**

30 As previously mentioned, cylindrical root sections are common due to their structural benefits, cheaper manufacturing and
31 easier transportation. However, regarding their aerodynamic behavior, bluff body like separation is inevitable and even at the
32 airfoils "blending" the root and outer rotor portion strong trailing edge separation is often present as a consequence of angle of
33 attack and in particular thickness induced adverse pressure gradients. Once the flow detaches, the centrifugal force transports
34 the fluid outwards in radial direction. During this motion, the fluid is subject to a Coriolis force accelerating the fluid again in
35 chord-wise direction. These rotational forces are responsible for a delay of stall compared to an equivalent two dimensional
36 section which is commonly known as the Himmelskamp effect (Himmelskamp, 1947). Experimental studies on the rotational
37 effects in the inboard sections have been conducted during NREL's Unsteady Aerodynamics Experiments by Schreck and
38 Robinson (2002) who compared surface pressure data and forces on the rotating blades against analogous stationary conditions.
39 During the MEXICO campaigns (Schepers and Snel, 2007), unsteady surface pressure as well as PIV measurements have been
40 conducted which served as validation basis for studies on stall delay in the hub region by Herráez et al. (2014) and Guntur
41 and Sørensen (2015). At TU Delft, experimental investigations using PIV were performed with focus on the root flow of a two
42 bladed model wind turbine (Akay, 2016). They could identify strong root vortices and high axial velocities in the root region.
43 Recently, Herráez et al. (2016) performed numerical simulations on the same rotor and characterized the Himmelskamp effect
44 as well as the origins of span-wise flow. A similar study has been conducted by Bangga et al. (2018) for the DTU 10MW rotor.

45 **1.2 Corner Flow Separation of Aerodynamically Shaped Junctions**

46 The wind turbine considered in the present study differs from conventional designs in the sense that it employs aerodynamically
47 shaped profiles down to the nacelle. This has the benefit that by using suitable flatback airfoils flow separation could be
48 generally eliminated under the isolated consideration of the rotor blade. It could be shown experimentally by Schreck et al.
49 (2013) that for a full scale turbine employing flatback profiles, flow separation was already negligible at $r/R = 0.14$. It must be
50 pointed out that the turbine considered in their study employed a cylinder like connection to the hub, i.e. there was no distinct
51 intersection length with the hub. In case that the airfoil shape is actually maintained towards the root, mutual interaction of
52 the wall boundary layers occurs. Indeed, the melded corner boundary layer is less rich in kinetic energy and therefore prone
53 to so called corner separation. This phenomena is typically found in the junction of the wing and the fuselage of transport
54 aircraft (Levy et al., 2014; Vassberg et al., 2008) or in turbo-machinery cascades (Knezevici et al., 2010). Corner flows are
55 highly influenced by a complex vortex system of primary and secondary vortices and have been investigated in great detail in
56 the experimental work addressing the effect of the horseshoe vortex (HSV) near the leading edge (see review paper of Simpson
57 (2001)) and more recent by Gand et al. (2015), focusing also on the trailing edge and analyzing the corner vortex. The latter

58 authors found particular evidence that the Reynolds stresses are not aligned with the mean shear tensor, which means that
59 anisotropy of turbulence is present in the region of corner flow separation. Numerically, this makes corner flow separations
60 very challenging to predict. From the NASA drag prediction workshops dedicated to aerodynamic predictions of a transport
61 aircraft (Vassberg et al., 2008) a huge scatter in the separation bubble size of the wing-fuselage junction was obtained in the
62 CFD predictions. Particularly the original version of the Spalart-Allmaras turbulence model (Spalart and Allmaras, 1992) has
63 shown to massively overestimate the bubble size. In order to reduce the model uncertainty for these kinds of flows, NASA very
64 recently initiated an own Junction Flow Experiment (Rumsey et al., 2016).

65 1.3 The Interacting Flow Fields of the Rotor and the Nacelle

66 **R1:G1** For a detailed analyses of the aerodynamic effects in the blade root region the isolated consideration of the the rotor
67 might not be adequate. It is obvious that depending on the shape of the nacelle, displacement effects alter the velocity field
68 acting on the blade. Moreover, the distribution of the bound circulation of the blade might be significantly changed the if there
69 is no air flowing around the root. Previous studies simulating the interacting flow fields of nacelle and the rotor were primarily
70 focused to understand the relationship between the free-stream wind speed and the velocity measured by the nacelle anemome-
71 ter. For this purpose the simulations in the work of Masson and Smaili (2006) resolved the detailed geometry of the nacelle and
72 modeled the rotor by an actuator disc approach. In the study of Zahle and Sørensen (2011) the same purpose was pursued but
73 using geometrically fully resolved blades, so that more local effects of the interacting flow fields of the blade and the nacelle
74 could taken into account. The authors showed a non-linear relationship of the local over-speed with wind speed. Regarding
75 the impact of the nacelle on the blade-aerodynamics of the root region, Johansen et al. (2006) redesigned a multi-megawatt
76 rotor by increasing the chord and twist in the root region and further incorporated an egg-shaped nacelle similar to the one
77 considered in the present study. Compared to a conventional rotor with cylindrical sections the thrust and power coefficient
78 could be locally increased for the inner 40% of the rotor. The relative improvement of the redesigned blade was found to be
79 higher compared to the additional benefit obtained by including the egg-shaped nacelle.

80 1.4 Improvement of the Aerodynamic Efficiency in the Blade Root Region

81 **R2:G1-a** A drag reduction of the cylinder like root region of conventional wind turbines blades can only be achieved by
82 diminishing the separated wake flow. Passive or active flow control strategies are promising technologies to favorably influence
83 the boundary layer. Passive solutions to mention are vortex generators (Baldacchino et al., 2018) that introduce kinetic energy
84 into the boundary, or Gourney flaps and spoilers, that redirect the flow by altering the Kutta-condition. More sophisticated
85 active systems featuring for example tangential blowing (Seifert et al., 1993; McCormick, 2000), or injecting momentum by
86 plasma actuators (Post and Corke, 2004) allow for specific control strategies without generating self-drag or noise of the device.
87 Although these active systems are promising in terms of stall alleviation, the industrial realization on wind turbine blades is
88 still pending, due the enormous technical complexity and high costs.

89 Despite their structural benefits and improved lift characteristics over conventional thick airfoils, flatback airfoils have the
90 drawback of generating significant drag in companion with blunt trailing edge noise. The reason for this is coherent vortex

91 shedding that leads to low pressure at the airfoil base. Hence, most of the drag reduction devices aim diminish the interaction
92 of the large scale vortex shedding with the airfoil base. This can be achieved by either breaking up the coherent structures using
93 serrated trailing edges, by hampering the von Kármán instability with splitter plates, or by means of cavities at the trailing edge
94 that shift the shedding vortices away from the airfoil base. An overview on these modifications can be found in Tanner (1975),
95 or Van Dam et al. (2008).

96 1.5 Scope and Objectives

97 **R2:G1-b** In order to improve the aerodynamic design of turbines in the root region and to stimulate research for such flow
98 control devices described above, a better understanding on the governing aerodynamic effects in the interference region of
99 the blade and the nacelle is extremely important. Particular emphasis in the present study is placed on identifying the driving
100 parameters for root separation with respect to the geometry of the nacelle. In course of this, a modification is introduced in
101 three categories with the purpose to improve the aerodynamic behavior in the root region by diminishing the separation caused
102 by the interfering structures. First, the relative nacelle thickness shall be varied, followed by a variation of the blade position
103 relative to the blade. Moreover, the effect of fillet-type geometry modifications in the blade-nacelle junction shall be discussed.
104 The most promising modifications are examined for off-design conditions, as well.

105 The next section gives an overview of all considered cases and modifications. In section 3 computational details on the flow
106 solver, settings and grids are provided. The results addressing the above defined objectives are discussed in section 4 and the
107 main conclusions are finally drawn in section 5.

108 2 Baseline Reference Turbine and Considered Modifications

109 **Author** The reference turbine used in this study shall feature the basic geometrical properties of a modern Enercon wind
110 turbine, namely the egg-shaped nacelle and flatback airfoils with large chord in the inboard region of the rotor. For reasons of
111 confidentiality, no original turbine could be used. Instead, a generic re-design of the Enercon E44 converter is employed. The
112 rotor diameter of this pitch regulated turbine is 44m, tilt and cone angles are set to zero. The induction, as well as the distri-
113 butions of the airfoil thickness and the solidity are representative for the root region of the industrial counterparts. However, in
114 the generic version the original airfoils have been replaced by open source DU and NACA airfoils. Further, the flatback trailing
115 edge segment is a purely generic design. The latter is responsible for a very high solidity of the rotor in the inner sections as
116 shown in the upper left graph of Fig. 1. The baseline nacelle geometry, is drop-shaped with $L = 8.1\text{m}$ length and a relative
117 thickness of 48%. The point of maximum thickness is at $x/L = 0.39$. The cut position between the rotating hub and the static
118 nacelle is at $x/L = 0.515$.

119 In order to isolate the governing fluid mechanical effects in the blade root region, different geometrical parameter sets will
120 be addressed which are summarized in Fig. 1. The baseline case described above will be referred to as *CaseT1.0*. The general
121 effects of the nacelle on the blade root flow shall be analyzed in comparison with an isolated rotor simulation (*CaseIsoRotor*).
122 From that, it shall be assessed, whether the isolated rotor assumption holds for rotors with high solidity.

123 In a second step, the relative thickness of the nacelle is varied from the baseline, being increased by a factor of 1.2 and 1.4,
124 obtaining the modifications *CaseT1.2* and *CaseT1.4*, respectively. For the latter case, the blade has been redesigned in the root
125 region involving a modified twist distribution (*CaseT1.4-twistMod*) which is shown in the upper left graph of Fig. 1. The twist
126 angle has been increased in the root region by around 2° and approaches the original distribution at $r/R = 0.36$.

127 The next modification category addresses a variation of the blade position relative to the nacelle. These modifications
128 are based on the nacelle with the largest thickness. *CaseT1.4-dXm04* denotes a blade shift in negative x -direction by 0.4m,
129 whereas a movement of the blade in rotational direction by $\Delta y = \pm 0.5\text{m}$ will be investigated in the cases *CaseT1.4-dYm05* and
130 *CaseT1.4-dYp05*, respectively. Additionally, the movement of the blade in direction of rotation shall be applied to the baseline
131 nacelle thickness, as well, yielding *CaseT1.0-dYm05*. The latter incorporates also a hump in the rear part of the suction side of
132 the blade.

133 Lastly, the geometry in the junction will be modified. In *CaseT1.0-rounded*, the junction line has been rounded by a constant
134 radius of $r = 0.4\text{m}$, whereas in *CaseT1.0-fairing*, the radius has been blended from $r = 0.2\text{m}$ at leading edge and on the
135 pressure side to $r = 0.85\text{m}$ at the trailing edge of the suction side.

136 3 Computational Details

137 3.1 Flow Solver and Numerical Settings

138 For the present study, the block-structured finite volume solver of the compressible Navier-Stokes equations FLOWer (Kroll
139 et al., 2000) by DLR has been used. This code is well suited for simulation of rotary wings, since the fluxes caused by
140 relative grid movements are taken into account. Thereby, the movements of the different components are realized with the
141 overset grid technique (Benek et al., 1986). At the authors' institute, FLOWer is continuously developed to improve simulation
142 capabilities of rotary wings, including high order schemes (Kowarsch et al., 2013), advanced turbulence modeling (Weihsing
143 et al., 2016), fluid structure interaction **Author** (Sayed et al., 2016; Klein et al., 2018), wake modeling (Weihsing et al., 2017),
144 and optimization for high performance computing (Letzgus et al., 2018). **R1:G4** The code has been extensively validated
145 for wind turbine flows among others during the MexNext projects providing accurate predictions of loads and wake measure-
146 ments (Schepers et al., 2012; Boorsma et al., 2018). During the EU-AVATAR project very competitive results were obtained in
147 the code-to-code comparisons for both uniform and complex inflow conditions (Sørensen et al., 2014, 2017).

148 In the present study, the discretization of the Euler fluxes is based on central differences with artificial dissipation according
149 to Jameson et al. (1981) using a $k4$ value of 128. Unless otherwise stated all simulations are performed steady state. For
150 unsteady simulations, a dual time stepping discretization according to Jameson (1991) is used. Viscous fluxes are discretized
151 by central differences. Regarding turbulence equations, FLOWer uses a fully implicit discretization and has implemented
152 several one- and two-equation models, as well as Reynolds stress closures. Unless otherwise stated all simulations use the $k-\omega$ -
153 SST model by Menter (1994). As discussed in Appendix A this model is able to predict primary vortices in junction flows and
154 showed good agreement with experiments regarding the extent of corner flow separation. **R1:G5-a** In order to entirely focus
155 on the impact of the geometric variations on the aerodynamics of the blade-root region and not to bring in additional model

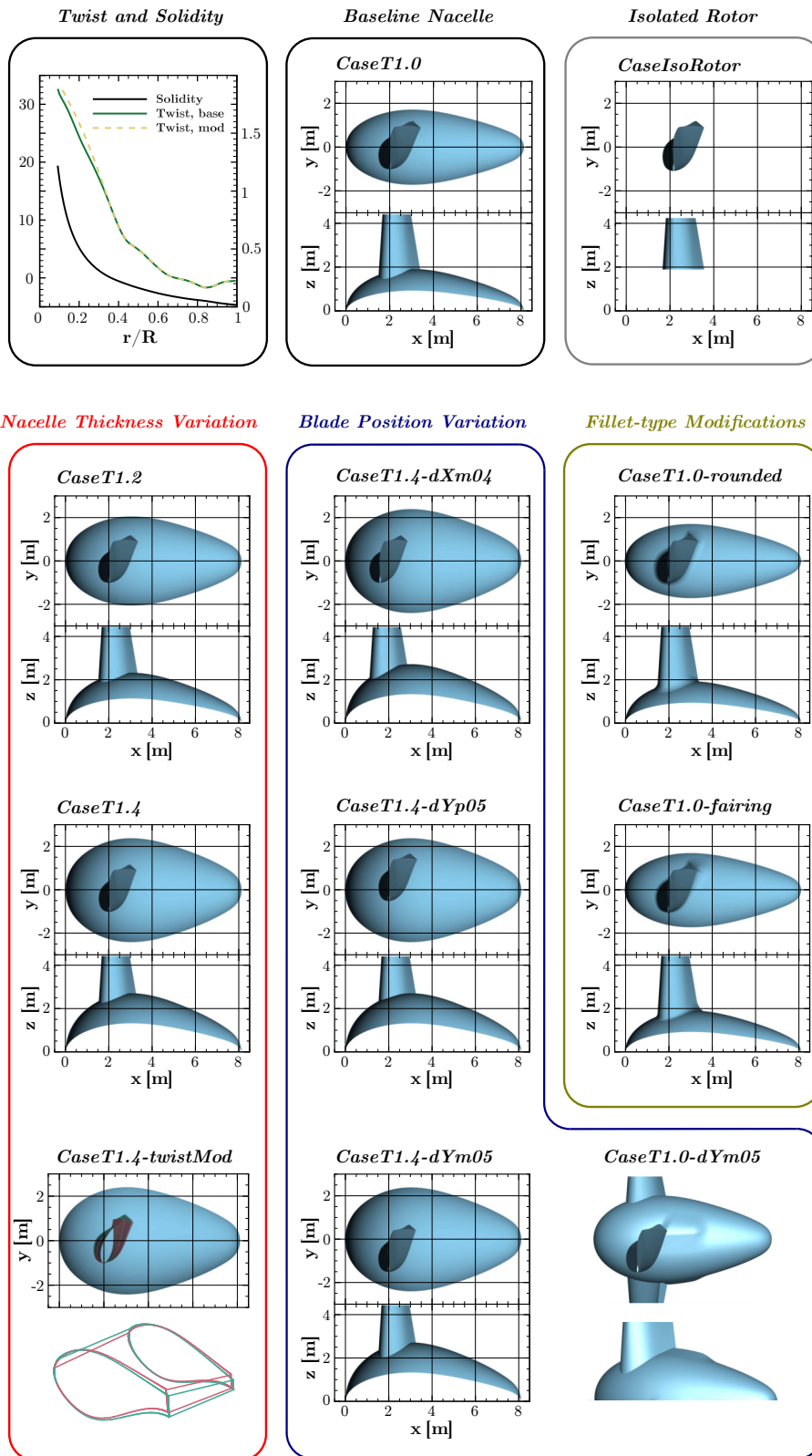


Figure 1. Considered geometric variations of the blade-nacelle region.

156 uncertainties due to boundary-layer transition, all simulations were conducted fully turbulent. It is pointed out that accurate
157 transition modeling in the considered junction region is non-trivial, since first no engineering model exists for the complex
158 transition mechanism within corner flows and second even outside of the corner-boundary-layer strong cross-flow effects pre-
159 vail. These are neither taken into account by the basic versions of for example the e^N -envelope method (Drela and Giles, 1987)
160 nor the widely used transport equation based local correlation model $\gamma-Re_\theta$ of Langtry and Menter (2009). To account for
161 cross-flow induced transition it shall be referred to the criteria based model of Arnal and Juillen (1987) or the recent extensions
162 of the $\gamma-Re_\theta$ model by Grabe and Krumbein (2014); Langtry (2015). Measurements as those of Zamir (1981) further suggest
163 an earlier transition to turbulence within corner boundary layers compared to corresponding flat-plate conditions. Neverthe-
164 less, in order to assess the first-order effects of transition on the results, transitional simulations are conducted for the baseline
165 rotor-nacelle configuration by employing the $\gamma-Re_\theta$ model as well as the envelope method of Drela and Giles (1987). For the
166 latter, the integral boundary layer properties are approximated according to Thwaites (1949). Due to its non-local formulation,
167 requiring a certain grid topology this model was only applied to the structure of the blade.

168 3.2 Grids and Boundary Conditions

169 The set-up for the simulations considers a periodic 120° segment of the flow problem and comprises the meshes for the rotor
170 blade and the nacelle which are embedded into a background grid. For topological reasons, the connections of the blade and the
171 nacelle in *CaseT1.0-rounded* and *CaseT1.0-fairing* use independent grids, which were created manually. All other grids were
172 generated automated by using appropriate *Gridgen* and *Pointwise* scripts, in order to obtain consistent results for all cases.

173 The polar background mesh extends 24 rotor radii upstream and downstream and 13 radii in radial direction, in order to
174 minimize effects from too close farfield conditions. It contains $448 \times 192 \times 128$ cells in stream-wise, radial and circumferential
175 direction. The grid spacing near the blade is $0.15\text{m} \times 0.3\text{m}$ in the rotor plane.

176 The mesh of the nacelle covers the whole blade grid and serves as refinement for the root region of the blade. The boundary
177 layer spacing is adapted on the Reynolds number assuring $y_p^+ \approx 1$. The geometric stretching factor is 1.1 and comprises 64
178 extrusion layers. The edge cell size of the latter is equally extended in radial direction up to $r/R = 0.25$ to adequately refine the
179 root region. In circumferential and stream-wise direction 320 and 168 cells are used, respectively.

180 The rotor blade is meshed in a two stage process. All cases use the same blade mesh for $r/R > 0.17$. This part was generated
181 using the *Automesh* script for rotor blade meshing developed at IAG. **R1:G2-a** **R2:G2-a** The chosen dimensions and spac-
182 ings were based on by the guidelines of the NASA drag prediction workshop Vassberg et al. (2010). The grid has a C-H
183 topology with a H-block extension of the tip using 288×168 cells in circumferential and radial direction, 64 cells on the
184 trailing edge and 60 cells in the wake. Again, a Reynolds number adapted first cell height was chosen and the near wall region
185 is resolved with 64 layers implying a growth rate of 1.11. In the second stage the blade mesh is extended in a conformal way
186 to each individual nacelle, including again 64 cells to refine the boundary layer region of the corner. It was preferred to choose
187 the more expensive H-type refinement of the corner boundary layer in order to avoid skewed cells would that have come when
188 wrapping the blade boundary around the corner. A detailed view of the surface meshes of the blade-nacelle intersection is
189 depicted in Fig. 2 showing the refinement in the junction as well as the overlapping of the grids.

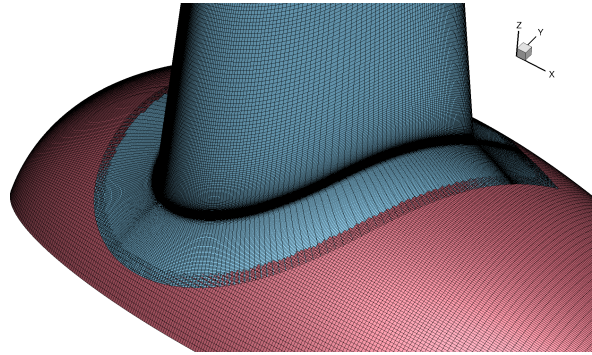


Figure 2. Surface grid of blade and nacelle in the corner region.

190 The configurations *CaseT1.0* and *CaseT1.4* have also been also simulated inviscid by the Euler equations. For those simu-
 191 lations, the grids have been modified by removing all boundary layer refinements which was necessary to achieve converged
 192 solutions.

193 As mentioned before, the rounding of the junction in the cases *CaseT1.0-rounded* and *CaseT1.0-fairing* does not allow for
 194 the previously described grid topology. A high quality grid with smooth cell distributions and as little as possible skewness
 195 could only be obtained by manually meshing the blade connector. This was conducted in O-type topology and is shown in
 196 Fig. 3.

197 The integration of the blade mesh into the nacelle mesh is illustrated in Fig. 4. In total, a typical set-up consists of around
 198 50M grid points.

199 Regarding the boundary conditions, the 120° domains are mapped periodically and the global domain boundaries carry
 200 far-field conditions. All surfaces are treated as no-slip walls (except for the additional Euler simulations of *CaseT1.0* and
 201 *CaseT1.4*). However, for the nacelle only the upstream part, the spinner, is rotating (Fig. 4 yellow colored patches), whereas
 202 the rear part (brown colored patches) stays fixed. This is achieved in the rotating grid setup without chimera or sliding mesh
 203 interfaces under exploitation of periodicity by subtracting the boundary velocity from the value of the interior cell, before
 204 setting the ghost layers.

205 3.3 Simulation Parameters and Operating Conditions

206 The main operating point considered is the one where the power coefficient is maximum, namely at a wind speed of $U_\infty = 10\text{m/s}$.
 207 In that case the tip speed ratio is $\lambda = 6$ and the blade pitch angle is zero. **R1:G6-a** Additionally to that point, the configura-
 208 tions *CaseT1.0* and *CaseT1.0-fairing* are analyzed for off-design conditions, as well by considering the further wind speeds,
 209 namely $U_\infty = 8; 12; 15\text{m/s}$.

210 **Author** Unless otherwise stated, all simulations at $U_\infty = 8; 10\text{m/s}$ have been performed steady state using 150,000 itera-
 211 tions. To verify convergence, selected simulations have been continued unsteadily by five more revolutions using a time step

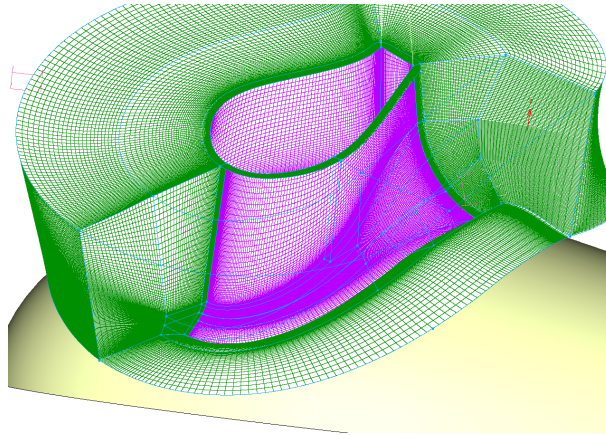


Figure 3. Grid topology for the fairing modification.

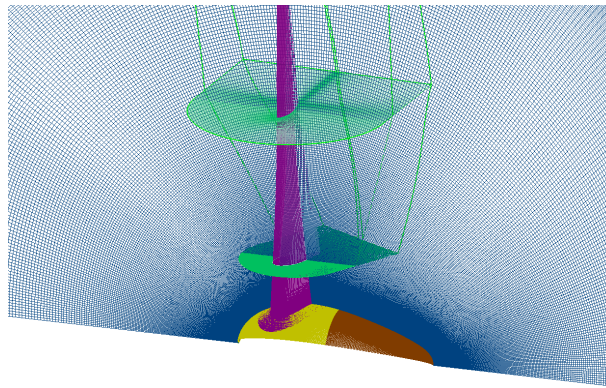


Figure 4. Integration of the blade mesh into the nacelle grid.

212 equivalent to 0.25° azimuthal movement. The averaged thrust coefficient of the fifth revolution deviated by less than 0.01%
 213 from the averaged value of the last 10,000 iterations of the steady-state solution. Besides of the integral values, the fluctuations
 214 of the radial force distribution in the root region were very small, since only shallow flow separation prevails. For these reason
 215 and the fact that computational time can be significantly reduced, all results presented in the following are based on steady-state
 216 solutions. **R1:G6-a** At the higher wind speeds $U_\infty = 12; 15 \text{ m/s}$ unsteady simulations were performed. Generally, it is clear
 217 that for detailed analyses addressing the unsteady behavior of the separated region, unsteady or even scale resolving simulation
 218 techniques should be applied.

219 4 Results

220 In this main section of the paper the simulation results of the different parameter groups shall be discussed, starting with a
221 comparison of the rotor including the nacelle with the isolated rotor and subsequently analyzing specific geometric variations,
222 as relative nacelle thickness, blade position and junction shape.

223 4.1 Influence of the Grid Resolution

224 **R1:G2-b** **R2:G2-b** In order to quantify the influence of the grid on the solution, the baseline grid described in section 3.2 shall
225 be compared to a refined version. The blade grid of the refined setup has the same topology as the one described before. It
226 contains 449×97 points in circumferential and wall-normal direction, 65 points on the trailing edge (the base of the flatback)
227 and a refined wake using 97 points. In span-wise direction the spacing corresponds to 1% local chord for the inner half of
228 the rotor. The blade-mesh itself contains 73M cells and the total setup incorporating a Cartesian refinement of the near-blade
229 region consists of 128M cells. The radial distributions of the sectional thrust and driving forces are cross-plotted for both reso-
230 lutions in Fig. 5 and 6. The close coincidence of the forces with almost collapsing curves indicate only a small influence of the
231 baseline grid on the solution. The integrated thrust force of the medium mesh is 0.139% lower compared to the refined grid.
232 For the integrated driving force the deviation is even smaller (0.007%), although the local deviation of the sectional loading
233 is slightly larger compared to the thrust force. Generally, the trends justify the dimensions and spacings for the grid described
234 in section 3.2. Nevertheless, it will be pointed out in section 4.6 that the effects obtained from the geometric modifications are
235 very local and their global benefits, e.g. obtained by *CaseT1.0-fairing* at low wind speeds are at the order of accuracy of the
236 used CFD framework.

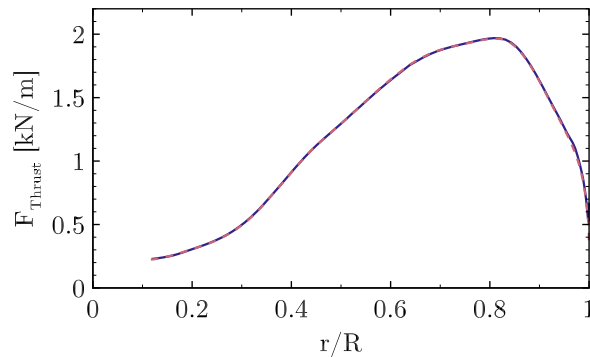


Figure 5. Influence of the grid resolution on the sectional thrust force. Standard medium mesh (dashed), refined grid (solid).

237 4.2 General Effects of the Voluminous Nacelle

238 To investigate the general effects of the nacelle on the blade root aerodynamics, the cases *CaseIsoRotor* and *CaseT1.0* are
239 compared. The flow field will be characterized showing relevant coherent structures, three-dimensional effects and stall are

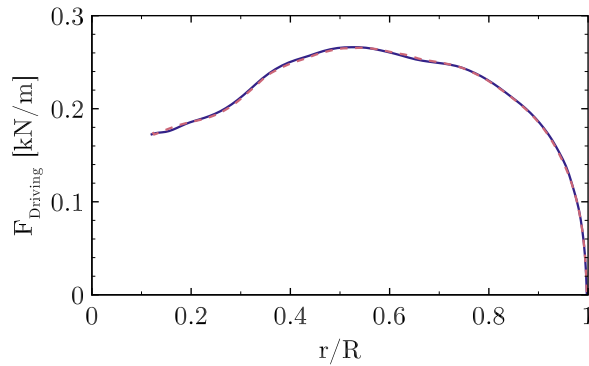


Figure 6. Influence of the grid resolution on the sectional driving force. Standard medium mesh (dashed), refined grid (solid).

240 going to be addressed. A digression will be made on the challenges of determining the AoA in the root region, before the effect
 241 of the nacelle on the aerodynamic coefficients will be summarized.

242 4.2.1 Vortical Structures

243 A first impression of the flow field can be gained by visualizing the dominant vortical structures using the Λ_2 criterion as well
 244 as the axial velocity distribution in the center cut, as shown in Fig. 7. For *CaseIsoRotor* the typical root vortices emerge from
 245 balancing the bound circulation at the root. The contour color of the iso-surface denotes the vorticity in y -direction and indicates
 246 the expected sense of rotation. The normalized axial velocity u/U_∞ in the slice shows a distinct jet through the hub which is
 247 responsible for the relatively high advance rates of the root vortices. Apart from those, no other relevant vortical structures are
 248 present that might indicate e.g. flow separation. Turning to *CaseTI.0*, shallow flow separation is visible on the suction side near
 249 the junction with the nacelle. Moreover, the dominant root vortex seen before has vanished. As will be discussed in more detail
 250 in Sec. 4.3, the vortices spiraling around the nacelle evolve from the two vortex legs of the HSV generated in the blade-nacelle
 251 junction. As the convecting velocity is significantly smaller than in *CaseIsoRotor*, these vortices which are counter-rotating
 252 come very close to each other until eventually, mutual interaction occurs. This results into the formation of three dimensional
 253 turbulent structures in the wake of the nacelle. The application of eddy resolving simulation techniques such as DES could
 254 bring further insight into these interaction phenomena (Weihing et al., 2016).

255 An effect that could be identified to enhance the generation of turbulent structures in the nacelle wake is the consideration
 256 of both, the rotating spinner and the steady rear part of the nacelle. In previous simulations during the project, where the
 257 entire nacelle had been rotated, these structures were almost absent (Kühn et al., 2018). In order to elaborate the effects of the
 258 boundary layer roll up and detachment in the rear part of the nacelle, isolated simulations of only the spinner and the static rear
 259 part of the nacelle have been conducted. The vorticity in stream-wise direction evaluated in the inertial frame of reference is
 260 shown in Fig. 8. The spinner is rotating in positive convention around the x -axis and accelerates the surrounding fluid that has
 261 no circumferential component far off the wall and therefore generates a negative x -vorticity above the spinner. At the interface

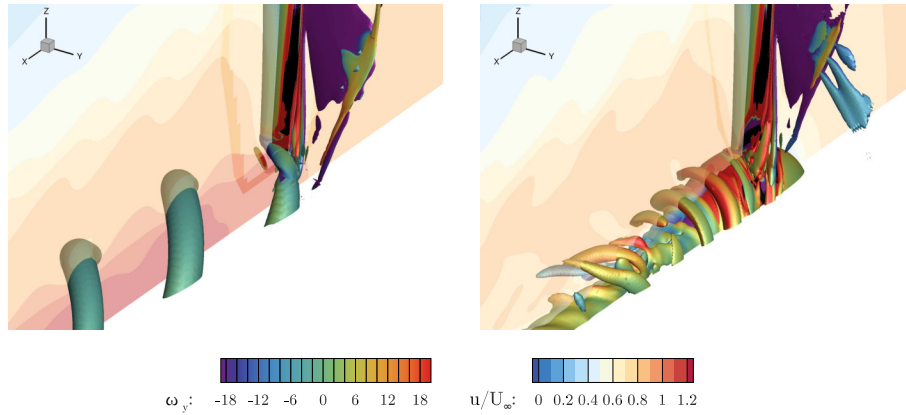


Figure 7. Vortices in the root area visualized by a Λ_2 iso-surface for *CaseIsoRotor* (left) and *CaseT1.0* (right). The vortices are colored by vorticity in y -direction. The $x = 0$ slice shows the normalized axial velocity component.

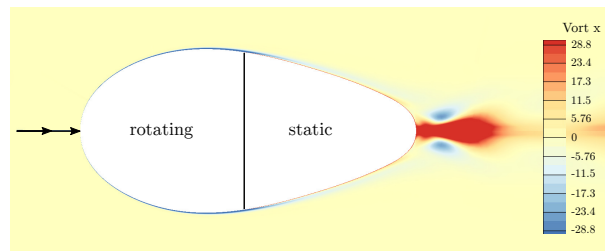


Figure 8. Vorticity in stream-wise direction in the boundary layer of an isolated nacelle including only the spinner and the static rear part.

262 to the non-rotating part of the nacelle this accelerated layer is suddenly retarded again, inducing a positive x -vorticity in the
 263 near wall region above the static surface. This results into a growing shear layer with positive circumferential velocity that
 264 eventually detaches from the surface and forms spiraling longitudinal vortices.

265 **Author** Turning to the mid-portion of the wake, a wavy velocity distribution can be observed for both cases at around
 266 $r/R = 0.3$. It is dedicated to trailed vorticity associated with the span-wise gradients of bound circulation. However, its strength
 267 appears to be significantly lower compared to the vorticity directly at the root.

268 4.2.2 Three Dimensional Flow and Separation in the Root Region

269 The extent of the flow separation as well as three dimensional flow patterns can be retrieved from the surface streamlines on
 270 the suction side of the blade shown in Fig. 9. For *CaseIsoRotor* there is no flow separation at the root. The slight radial flow
 271 component is dictated by the root vortex rolling from the pressure to the suction side. This is also reflected in the velocity
 272 contours in a slice $x = 2\text{m}$ just cutting the leading edge of the blade. The plotted velocity contours indicate the deviation
 273 velocity \tilde{w} describing the vertical motion relative to the ideal circular path. Negative values as for *CaseIsoRotor* mean that the
 274 streamlines stem from further outboard than assumed from kinematics.

275 By including the nacelle, its displacement effect evokes an effective inclination angle in that plane which results in a positive
 276 deviation component \tilde{w} . For the inner 20% of the rotor it is more than 0.5m/s. The negative values at around $r/R = 0.3$ are
 277 subject to the induction of the mid-trailing vortex described in the previous section. At the inboard sections, a clear flow
 278 separation evolves from the junction with the nacelle. It spreads in radial direction and realigns with chord direction at about
 279 $r/R = 0.17$. When focusing on the pressure contours, those are mostly parallel for *CaseIsoRotor* except for the very inboard
 280 region that is directly influenced by the root vortex. For *CaseTI.0* significant curvature of the isobars is visible in the area
 281 covered by the dividing streamline. As part of the separation process it can be observed that the angle of the shear stress vector
 282 turns relative to the pressure gradient, indicating a complex interaction of centrifugal-, Coriolis- and pressure forces as drivers
 283 of the three dimensional flow. Due to the very high solidity of the blade, which varies between $c/r \approx 1.1$ at $r/R = 0.1$ and
 284 $c/r \approx 0.5$ at $r/R = 0.2$ a high impact of three dimensional effects can be expected.

285 To estimate the share of these forces acting in the yz -plane, the simple balance can be written for a rotation around the x -axis
 286 as

$$287 \quad 0 = - \left(\frac{\partial p}{\partial y} \right) + \rho \left(\Omega^2 y \right) + \rho \begin{pmatrix} 2\Omega w \\ -2\Omega v \end{pmatrix}, \quad (1)$$

288 where the second and third term denote the centrifugal and Coriolis forces, respectively. The velocity components v and w are
 289 acting in chord- and span-wise direction relative to the moving blade rotating with Ω . As can be directly seen in the equation,
 290 by definition, the Coriolis force vanishes when the velocity is tangential to the rotation, hence it is expected to be small
 291 directly above the nacelle. For the balance in span-wise direction the Coriolis force changes sign with respect to the chord-
 292 wise velocity component, acting inward for attached flow and outward for separated flow and particularly vanishing near the
 293 dividing streamline. In contrast to that, the centrifugal force component in span-wise direction is simply proportional to the
 294 z -coordinate. Therefore, at the point of separation, the only force to hold the fluid particle on the path of rotation would be a
 295 span-wise pressure gradient dp/dz . The latter can be derived from the radial pressure distribution plotted in Fig. 10 for different
 296 chord-wise sections. At $x_c/c \approx 0.5$ which is close to the separation point, there is virtually no span-wise pressure gradient.
 297 Hence, the fluid moves outward solely due to the centrifugal force. It would accelerate towards the tip due to the increasing
 298 centrifugal force with z , if there was no Coriolis force. As the outward movement is naturally connected with a positive w -
 299 component it deflects the flow in chord-wise direction and once there is a chord-wise component and a positive v -velocity, an
 300 additional inward deflection is induced, until eventually, the flow realigns with the chord direction. The transport of separated
 301 fluid in outward direction is known as centrifugal pumping (Lindenburger, 2003) and allows for a pressure recovery also in radial
 302 direction, where the adverse pressure gradient is smaller compared to the chord-wise direction (see e.g. $x_c/c \approx 0.85$ in Fig. 10).
 303 For equivalent two-dimensional conditions, most of the pressure recovery would therefore occur in the turbulent mixing of
 304 the airfoil wake. This explains the commonly observed stall delay of a rotating blade compared to equivalent two-dimensional
 305 conditions (e.g. (Lindenburger, 2003), (Snel et al., 1993)).

306 The current investigations supports the studies of Du and Selig (2000), Lindenburger (2003) and recently Herráez et al. (2016)
 307 who explained the radial flow with the centrifugal force as well. In this study, the significantly higher blade solidity caused by
 308 the flatback airfoils with large chord is seen to be responsible for a significant impact of three dimensional effects.

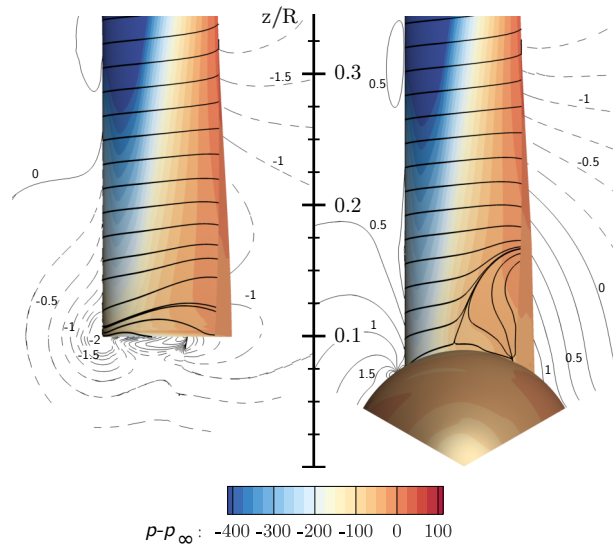


Figure 9. Visualization of three dimensional flow in the root region using streamlines and pressure contours on the suction side of the blade for *CaseIsoRotor* (left) and *CaseT1.0* (right). The contour lines indicate the "deviation velocity" \tilde{w} from to the ideal kinematic value $-\Omega y$ in a slice at $x = 2\text{m}$.

309 Turning to the front part of the blade, it can be shown that following Bernoulli's principle, the static pressure decreases in
 310 radial direction due to the increasing dynamic pressure of the inflow. This results in an outward pressure gradient. However,
 311 it is not believed to be the only reason for the slight radial component of the streamlines near the leading edge, since first,
 312 the pressure gradient is balanced by the Coriolis force in negative z -direction and further, a strong acceleration in chord-wise
 313 direction prevails, which is in accordance with Schreck and Robinson (2002). As described in more detail in Sec. 4.4 it is more
 314 related to an effective sweep angle of the blade relative to the inflow direction. This is again an effect of the high solidity of
 315 the blade which results into a leading edge shift relative the rotational axis in y -direction. Comparing the pressure levels, the
 316 suction peak is significantly reduced in *CaseIsoRotor* due the induction of the root vortex over the whole inner portion of the
 317 rotor.

318 4.2.3 Determination of the Angle of Attack in the Root Region

319 In order to assess the aerodynamic performance of the individual blade sections in terms of aerodynamic coefficients and to
 320 utilize that information in two-dimensional approaches such as BEM or for airfoil design tools, it is essential to accurately
 321 determine the angle of attack (AoA) from the three-dimensional flow field. However, it is clear that its extraction is far from
 322 trivial as wind velocity and rotation are superimposed by the induction of the wake and the bound circulation of the blade
 323 itself. Most approaches such as those compared by Rahimi et al. (2017) aim to eliminate the effect of the bound circulation.
 324 They provide very similar results at the mid-board blade sections. However, at the blade tip and root, major differences were
 325 observed. For the root region those were explained by the massive flow separation in the wake of the very thick airfoils. As for

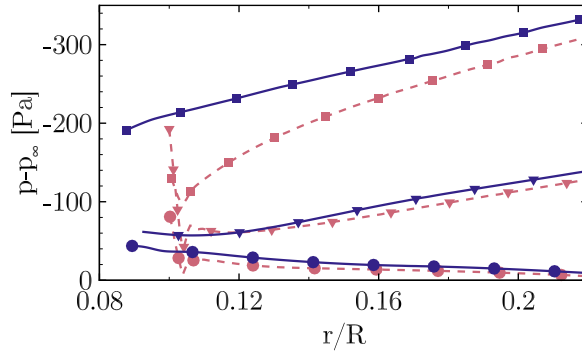


Figure 10. The effect of the nacelle on the radial pressure distribution on the suction side, at $x_c/c \approx 0.1$ (squares), $x_c/c \approx 0.5$ (triangles) and $x_c/c \approx 0.85$ (circles) for *CaseIsoRotor* (dashed), *CaseT1.0* (solid)

326 the present rotor flow separation is only shallow, or even absent, differences between the AoA extraction methods are strongly
 327 linked to the effects of the root vortex system, instead of the dominant vortex shedding caused by massive flow separation.
 328 Therefore, two AoA evaluation methods shall be compared with focus on the root region.

329 The first is the *Reduced Axial Velocity* method (RAV) of Johansen and Sørensen (2004), which eliminates the bound circula-
 330 tion by azimuthally averaging the axial velocity for each radial section. Secondly, the *Line Average* method of Jost et al. (2018)
 331 is applied which averages the velocity vector along closed lines around the blade. Both methods are sketched in Fig. 11. As
 332 seen in the figure, for the *Line Average* method, the extraction of the velocities occurs along circles that are curved about the
 333 rotational axis with a radius corresponding to the location of the quarter-chord point in each section, being centered around the
 334 latter. The radius of the projected circle is one chord length. By averaging the planar velocity vector along these curves, Jost
 335 et al. (2018) could show that the effect of bound circulation can be eliminated resulting in the pure inflow conditions for that
 336 section. The evaluation of the AoA in Fig. 12 shows a lower AoA near the tip for the *Line Average* method, since it locally
 337 accounts for the induction of the tip vortex, which is smeared out by the azimuthal averaging in the RAV method (c.f. Fig. 11).
 338 Accounting for the local induction is a desired feature of the *Line Average* method and has shown to accurately reconstruct
 339 the inflow conditions for corresponding two-dimensional analyses of the near tip region (Jost et al. (2018) and Rahimi et al.
 340 (2017)). Near the blade root, the AoA predicted by the *Line Average* method is higher than obtained by the RAV method. The
 341 reason for this is that the azimuthal arc length for averaging the axial velocity in the RAV method is now smaller compared to
 342 averaging over the circular line of the *Line Average* method (c.f. Fig. 11). Particularly, the acceleration of the curving stream-
 343 lines upstream of the airfoil are not considered in the RAV method. This effect becomes particularly important for *CaseT1.0*,
 344 where both the blade and the nacelle displace the flow and therefore accelerate it upstream of the leading edge which is the
 345 reason why the increase of AoA is more pronounced for the case including the nacelle. Additionally, it must be noted that for
 346 the RAV method the root vortex is occupying about 40% of the downstream arc length, being more concentrated for the *Line*
 347 *Average* method. As a consequence the latter predicts a higher axial velocity in the rotor plane compared to the RAV method

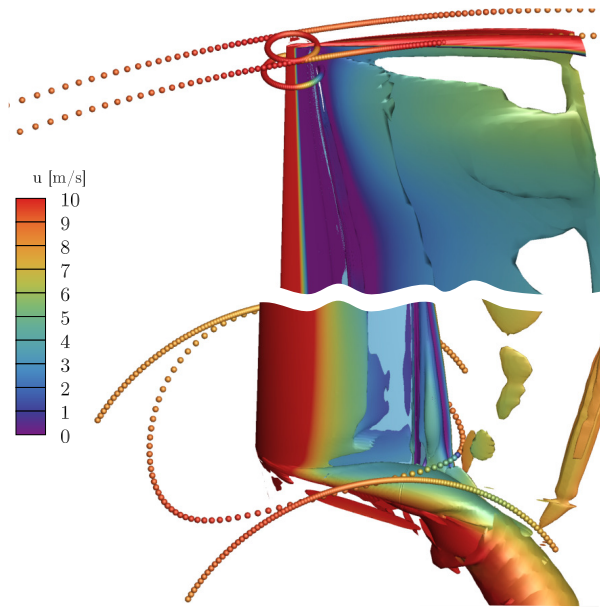


Figure 11. Extraction of axial velocity in the inertial frame near the tip and the root using the *Reduced Axial Velocity* method (Johansen and Sørensen, 2004) (circular arcs) and the *Line Average* (closed curves) (Jost et al., 2018) method.

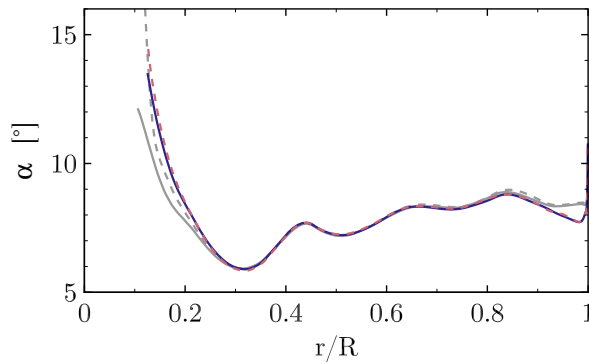


Figure 12. Radial distribution of the angle of attack for *CaseIsoRotor* (dashed) and *CaseTI.0* (solid) using the RAV method (gray) and the *Line Average* method (colored).

348 (c.f. Fig. 13), and therefore a higher AoA. In the remainder of the paper, the AoA will be evaluated with the *Line Average*
 349 method.

350 4.2.4 The Effect of the Nacelle on Aerodynamic Coefficients

351 The pressure coefficient C_p is compared for the configurations *CaseIsoRotor* and *CaseTI.0* as well as two-dimensional airfoil
 352 simulations in a slice at $r/R = 0.136$ (Fig. 14). Following Bernoulli's principle, the pressure coefficient has been normalized

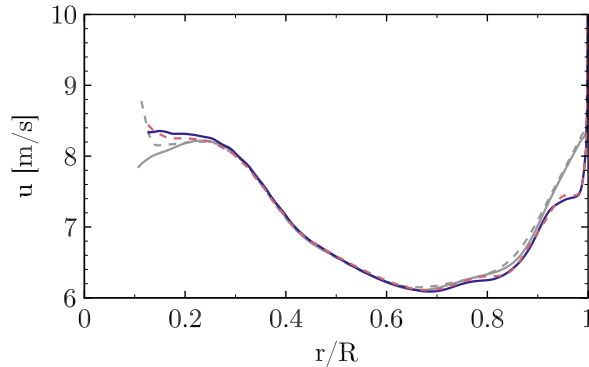


Figure 13. Radial distribution of the axial inflow velocity for *CaseIsoRotor* (dashed) and *CaseTI.0* (solid) using the RAV method (gray) and the *Line Average* method (colored).

353 using the maximum dynamic pressure that can be exploited from the kinematics $q_{\infty,rot} = \rho/2 (U_{\infty}^2 + (\Omega r)^2)$. In accordance
 354 with the independence principle of swept wing theory, only the component normal to the leading edge is relevant for the
 355 aerodynamic properties. Therefore, the z -coordinate has been used as radius for the circumferential component in the dynamic
 356 pressure. It should be noted that one could also account for the induction and sweep angle effects, but then the reference
 357 state becomes rather ambiguous and difficult to compare among the different cases. Hence, it will be accepted that with the
 358 kinematic normalization, C_p might differ from a value of 1 in the stagnation point and it will be attempted to explain this with
 359 the dominant physical effects.

360 As already discussed in the previous section, the induction of the root vortex significantly diminishes the suction peak for
 361 *CaseIsoRotor*. For the rotor including the nacelle, the pressure level is lower for the entire suction side of the blade, particularly
 362 in the front part of the airfoil. Hence higher lift and lower pressure drag can be expected for *CaseTI.0*. At the pressure side the
 363 differences are small. Only directly at the stagnation point a higher pressure coefficient is obtained by *CaseIsoRotor*, which is
 364 slightly > 1 . This can be explained by the negative \tilde{w} contours (Fig. 9) which indicate that the fluid hitting the airfoil is actually
 365 stemming from further outboard compared to its ideal kinematic path. Therefore, fluid of higher momentum is transported
 366 towards the leading edge part of the airfoil, yielding a higher stagnation pressure than assumed for normalization. By contrast,
 367 in *CaseTI.0* the stagnation pressure is lower than estimated, due to an effective inclination angle which pushes fluid from the
 368 inboard sections to the reference cut. Finally, focusing on the aft chord region, it can be confirmed that also for *CaseTI.0* the
 369 cut is outside of recirculation and that pressure has almost recovered by the centrifugal pumping mechanism.

370 To further assess the three-dimensionality of the flow in the hub region, a comparison shall be drawn with two-dimensional
 371 simulations. Those were performed at the angles of attack $\alpha = 10.2^\circ$, 13° and 14° , where the first corresponds to the extracted
 372 value from the RAV method for *CaseTI.0*, the second corresponds to a slightly higher value than obtained from the *Line Aver-*
 373 *age* method which was 12.3° for *CaseTI.0* and the last is close to the AoA obtained for increased nacelle thicknesses as shown
 374 in Sec. 4.3. The 2D results at $\alpha = 10.2^\circ$ reveal no separation. At $\alpha = 13^\circ$ shallow trailing edge separation is present which
 375 moves forward to $x_c/c \approx 0.6$ at $\alpha = 14^\circ$. Compared to the 3D results, *CaseIsoRotor* shows a completely different behavior

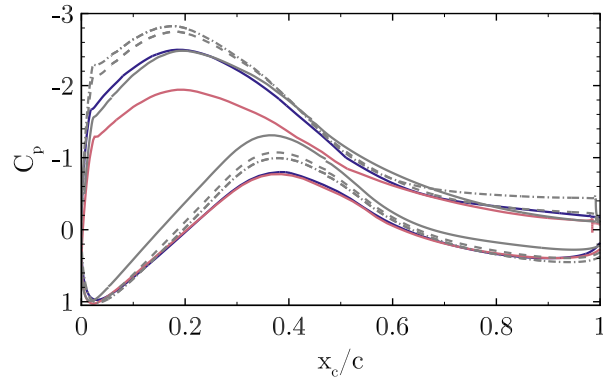


Figure 14. Pressure coefficient C_p at $r/R = 0.136$ for *CaseIsoRotor* (solid red) and *CaseT1.0* (solid blue) compared with two-dimensional simulations (gray lines): $\alpha = 10.2^\circ$ (solid), $\alpha = 13^\circ$ (dashed), $\alpha = 14^\circ$ (dashed dot).

376 which cannot be mimicked by any 2D simulation. At a first glance the level of the suction peak of *CaseT1.0*, suggests that
 377 $\alpha = 10.2^\circ$ would be a good approximation. However, it must be pointed out that when scaling the stagnation pressure in the
 378 3D results to a value of 1, the distribution increases a bit. Moreover, the visual inspection of corresponding streamline plots
 379 compared in that section (not shown here) clearly suggested a larger AoA than 10° . The distribution in the adverse pressure
 380 regime is very similar to the 2D results at $\alpha = 13^\circ$ implying a similar boundary layer stress. The differences on the pressure
 381 side are also markedly, showing distinctly stronger suction peaks predicted by any of the two-dimensional simulations. These
 382 examinations emphasize the strong three-dimensionality of the flow in the root region of the present rotor and point out that
 383 the application of two dimensional polars in BEM without correction models must be treated with caution, even if there is no
 384 massive flow separation.

385 **R2:G4-a** The radial distribution of the circulation Γ plotted in Fig. 15 a), as well as the lift coefficient (Fig. 15 b)) confirm
 386 the improvement of the aerodynamics in the root region by including the nacelle which effectively diminishes the harmful in-
 387 ductive effect of the root vortex. This is also reflected in a reduction of the drag coefficient (Fig. 15 c)) by up to 100 drag counts,
 388 although *CaseT1.0* involves flow separation. The region of influence of the nacelle extends to $r/R = 0.35$. By integrating the
 389 driving force within that range, this results in a higher torque of around 20% for that portion.

390 4.3 The Impact of the Relative Nacelle Thickness

391 From the previous section it was concluded that an improvement of the aerodynamic properties in the inner portion of the rotor
 392 could be obtained by taking into account the nacelle. The attenuation of the root vortex by the nacelle diminished induced drag
 393 and increased lift. In this section the effects of the relative nacelle thickness will be analyzed. The thickness has been increased
 394 in two steps by a factor of 1.2 and 1.4 (*CaseT1.2*, *CaseT1.4*), respectively.

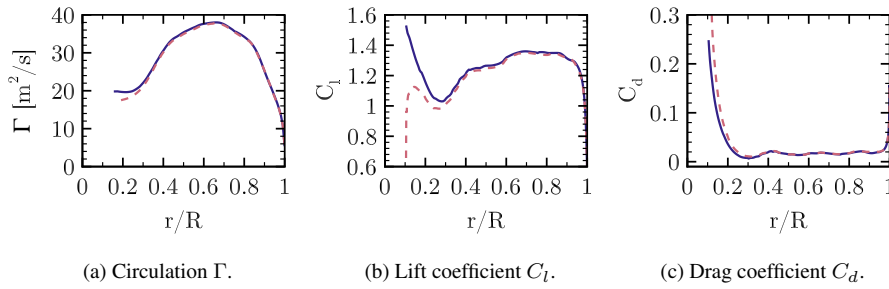


Figure 15. Distribution of aerodynamic coefficients along the blade radius for *CaseIsoRotor* (dashed) and *CaseT1.0* (solid).

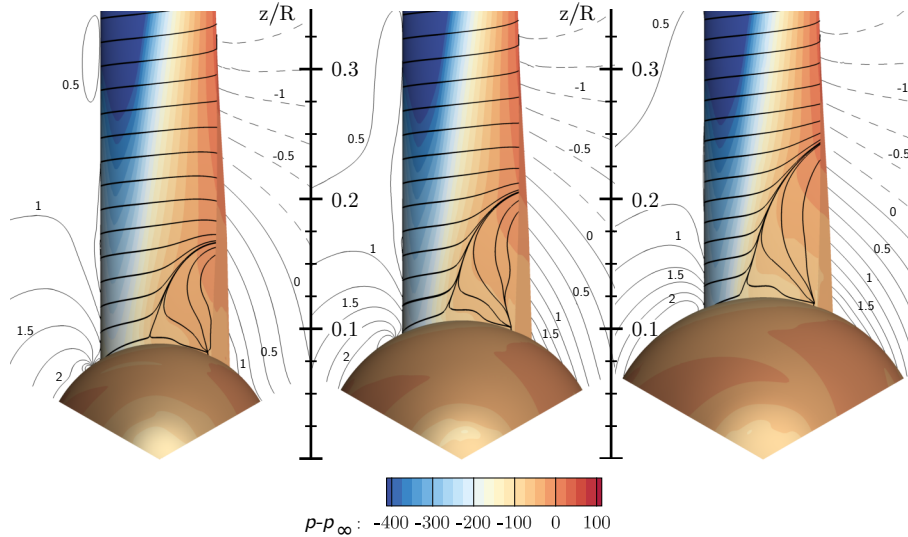


Figure 16. Visualization of three dimensional flow in the root region using streamlines and pressure contours for *CaseT1.0* (left), *CaseT1.2* (mid) and *CaseT1.4* (right). The contour lines are plotted for a slice at $x = 2m$ and indicate the velocity $\tilde{w} = w - \Omega y$ which denotes the deviation from the ideal kinematic path.

395 4.3.1 The Effect on Flow Separation and its Driving Parameters

396 The resulting surface streamlines and pressure contours shown in Fig. 16 indicate growing flow separation with increasing
 397 nacelle thickness. **Author** The extent of the separation measured from the point of maximum thickness of the nacelle to
 398 the radial position of reattachment increases from $0.08R$ over $0.10R$ to $0.12R$ for the configurations *CaseT1.0*, *CaseT1.2* and
 399 *CaseT1.4*, respectively. With increasing thickness of the nacelle the separation line moves forward by around 15%. In addition,
 400 it can be noted from the "deviation velocity" \tilde{w} that the inclination of the inflow increases with larger relative nacelle thickness.

401 The comparison of the pressure coefficient in the reference cut at $r/R = 0.136$ (Fig. 17) confirms the increasing separation
 402 for larger nacelle thicknesses. The distinct pressure plateau already suggests a loss of aerodynamic efficiency due to flow

403 separation. The radial pressure gradients in the front part of the blade (Fig. 21) show that greater relative nacelle thickness
 404 increases the suction force due to the displacement effect of the nacelle. Also, in the mid- and aft- chord region the initial
 405 pressure level is lower, which increases the radial adverse pressure gradient. Since separation inherently alters the pressure
 406 distribution, it is difficult to distinguish between the causes and effects of the flow separation. In order to assess whether stronger
 407 adverse pressure gradients develop by increasing the nacelle thickness, inviscid reference simulations have been performed for
 408 *CaseT1.0* and *CaseT1.4*. The interaction of the inviscid pressure fields is visualized in Fig. 19 using vectors of static pressure
 409 acting on the surfaces. For *CaseT1.0* the flow accelerates moderately in the junction, reaching the minimum of pressure at about
 410 $x_c/c \approx 0.5-0.6$. It can be shown that the adverse pressure gradient imposed by the nacelle is slightly shifted behind the one from
 411 the blade. Turning to *CaseT1.4* this is not the case anymore, as the suction peaks fairly coincide at around $x_c/c \approx 0.3-0.4$.
 412 Downstream of that point the adverse pressure gradient markedly increases compared to *CaseT1.0* and is accordingly devolved
 413 on the blade as shown in Fig. 17. In *CaseT1.0* the inviscid and viscous suction peaks closely coincide, whereas a significantly
 414 higher peak prevails for the inviscid simulation of *CaseT1.4*. This clearly suggests that the boundary layer of the associated
 415 viscous case is increasingly loaded with larger nacelle thickness. From these investigations it can be concluded that a reduction
 416 of separation might be achieved by decoupling the interfering pressure gradients of the nacelle and the blade as it is typically
 417 conducted for winglets.

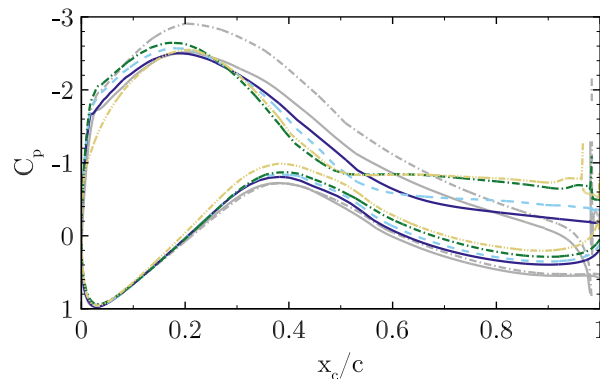


Figure 17. Airfoil pressure distribution at $r/R = 0.136$ for *CaseT1.0* (solid), *CaseT1.2* (dashed), *CaseT1.4* (dashed dot) and *CaseT1.4-twistmod* (dashed dot dot). The gray curves denote corresponding inviscid simulations.

418 A second aspect, that turned out to be crucial for the development of the flow separation is directly linked to the vortex system
 419 evolving in the junction of the blade and the nacelle. The motivation for a detailed look into that came across, by analyzing
 420 AoA behavior with respect to the different nacelle geometries (Fig. 22), where an increasing AoA of around two degrees could
 421 be observed from *CaseT1.0* to *CaseT1.4*. In the beginning of the investigations, it was assumed that the increasing separation
 422 was the primary effect of the larger AoA. Therefore, it was attempted to re-design the blade in order to compensate the AoA
 423 leading to *CaseT1.4-twistMod* which employs an increased twist angle in the blade root region of around 2.0° (see Fig. 1).

424 **Author** Except for the sections very closely to the root, the AoA could be effectively reduced. Very inboard, the aerodynamic

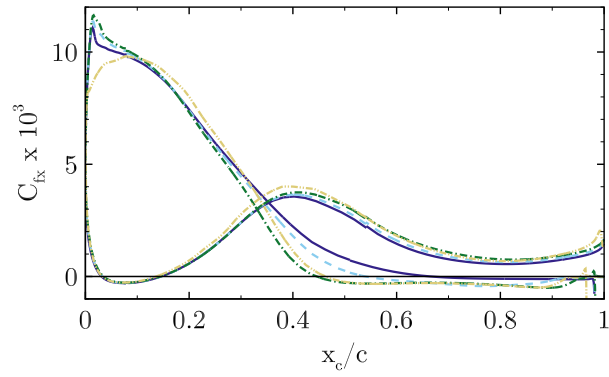


Figure 18. Airfoil skin friction distribution at $r/R = 0.136$ for for *CaseT1.0* (solid), *CaseT1.2* (dashed), *CaseT1.4* (dashed dot) and *CaseT1.4-twistmod* (dashed dot dot).

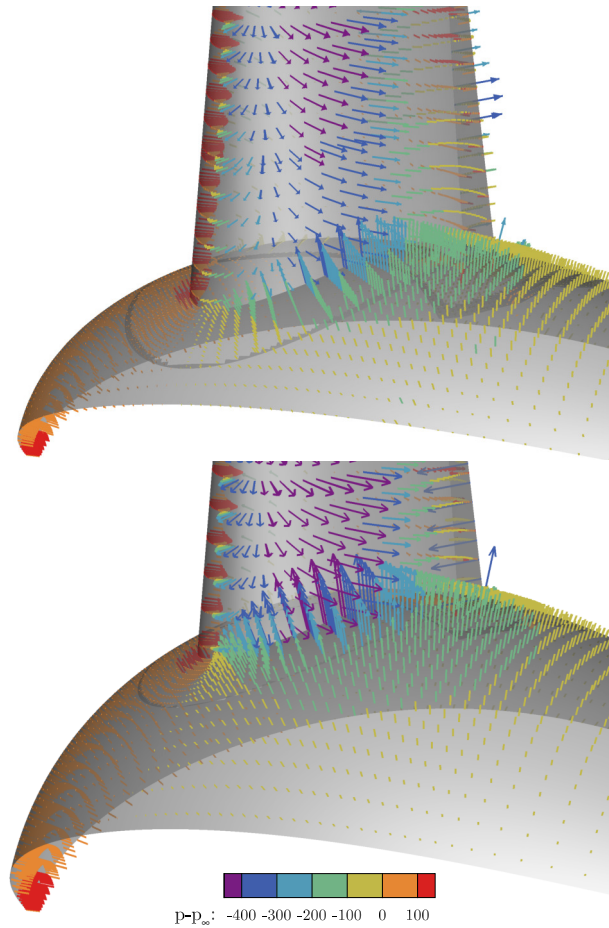


Figure 19. Inviscid pressure distribution in the junction region of blade and nacelle for *CaseT1.0* (upper) and *CaseT1.4* (lower).

425 behavior is obviously strongly non-linear. Due to the twist modification the flow is redirected which causes an increase in the
426 axial velocity. The latter is detected by the *Line Average* method which evaluates a lower AoA reduction than expected by the
427 geometric twist modifications. In turn the RAV method was closer to the kinematic value. Nevertheless, when looking at the
428 pressure distribution in Fig. 17, the suction peak is significantly reduced. The evaluation of chord-wise skin friction (Fig. 18)
429 shows that the separation remains almost the same as for *CaseT1.4*, so it could be concluded that flow separation is not directly
430 affected by an AoA induced adverse pressure gradient on the blade.

431 It is more seen to depend on the interacting boundary layers in the junction region of the blade and the nacelle. To shed
432 more light into that, the emerging vortices in the junction are visualized in Fig. 20 for *CaseT1.0* and *CaseT1.4* using volume
433 streamlines colored by vorticity in the local direction of the velocity vector. Additionally, volume cuts are placed normal to
434 the blade at the leading edge and at $x_c/c = [0.25; 0.5; 0.75]$, respectively. The horseshoe vortex (HSV) is clearly visible and
435 seems to be generated in the stagnation region, when the boundary layer of the nacelle approaches the blade. It is rolling
436 inward and its size and strength could be observed to increase with larger nacelle thickness. This behavior is consistent with
437 Simpson (2001), who reports a stronger HSV with increasing AoA. Gand et al. (2015) showed experimentally that the onset of
438 corner separation is delayed by a stronger HSV, since fluid of higher momentum is pushed into the blade boundary layer. This
439 beneficial inductive effect, likewise depends on the distance to the blade. By increasing AoA the suction side leg departs from
440 the blade and is further deflected by the Coriolis force.

441 Directly inboard of the HSV, the counter rotating corner vortex (CV) evolves from the stagnation point and closely follows
442 the juncture of the blade and the nacelle. Its production depends on the velocity gradients of the interacting boundary layers
443 and its strength was observed to increase for the larger relative nacelle thickness in the adverse pressure gradient region. In
444 contrast to the HSV, the CV remains aligned with the junction. Due to its sense of rotation it seems to be responsible to deform
445 the near wall velocity profile on the blade and to pull the boundary layer flow away from the wall. For all cases it was observed
446 that the recirculation area was initiated from the streamline originating in the CV. Thus, high vorticity in the CV might be an
447 important driver for the whole separation process. In particular it should be noted that the configuration with modified blade
448 twist (*CaseT1.4-twistMod*) revealed quasi identical values for the CV strength as *CaseT1.4* and showed the same amount of
449 separation.

450 Hence, the second strategy to diminish the detrimental flow separation could be a relief of the mutual loading of the
451 boundary layers of the blade and the nacelle, in order to influence the CV strength and propagation. This aspect shall be
452 investigated in Sec. 4.5.

453 4.3.2 The Effect of the Relative Nacelle Thickness on Aerodynamic Coefficients

454 **R2:G4-b** To summarize the effects of the relative nacelle thickness on the aerodynamic coefficients in the root region which
455 are plotted in Fig. 22, those confirm the degradation of aerodynamic performance due to stronger flow separation in the in-
456 board region, where lift mostly decreases and particularly drag increases. The AoA increases with the nacelle thickness, since
457 reduced lift decreases axial induction, which yields higher axial velocities in the rotor plane. The decrease of aerodynamic
458 efficiency is most prominent for the inner 25% of the rotor radius. It should be noted that there was no measurable benefit for

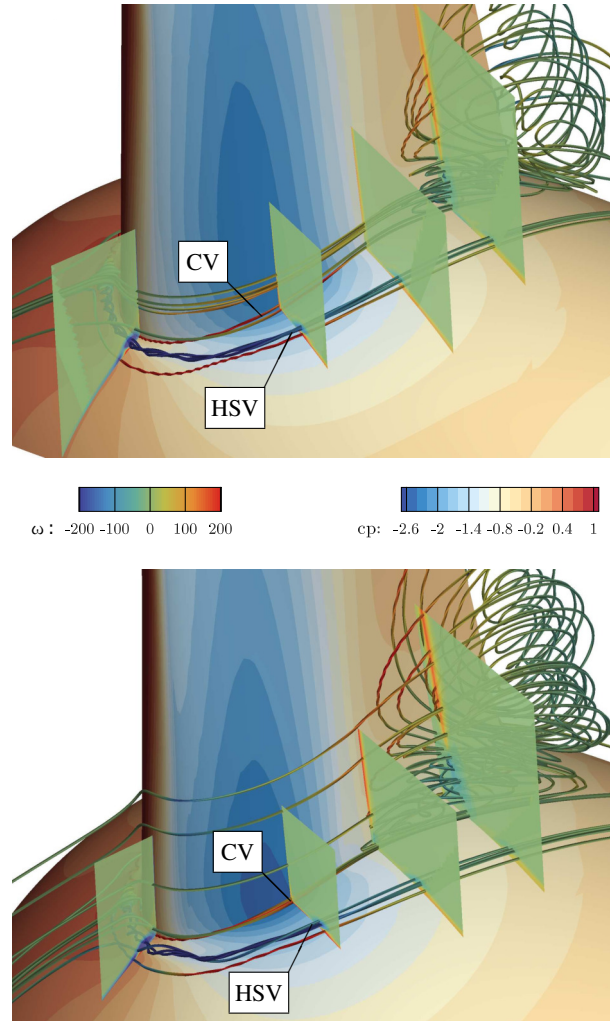


Figure 20. Vortex system in the junction of blade and nacelle for *CaseT1.0* (upper) and *CaseT1.4* (lower).

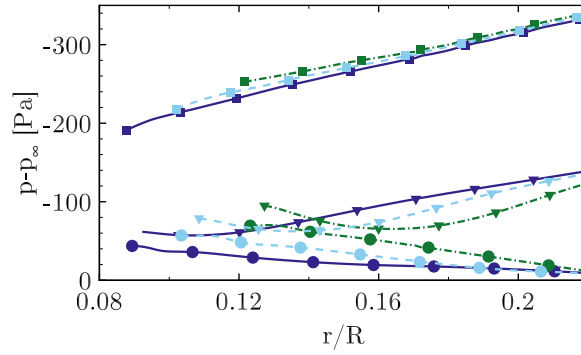


Figure 21. The effect of the nacelle thickness on the radial pressure distribution on the suction side, at $x_c/c \approx 0.1$ (squares), $x_c/c \approx 0.5$ (triangles) and $x_c/c \approx 0.85$ (circles) for *CaseIsoRotor* (dashed), *CaseT1.0* (solid), *CaseT1.4* (dashed dot).

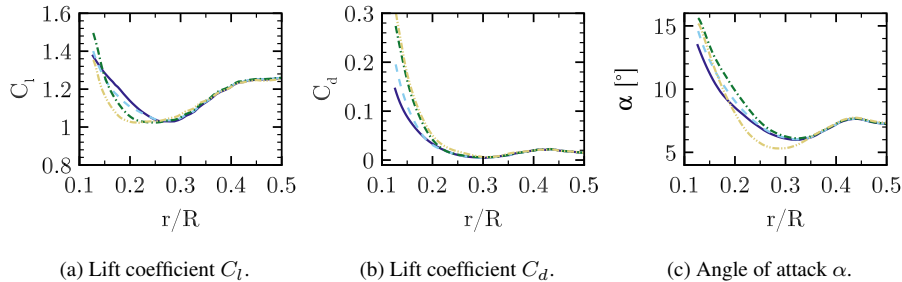


Figure 22. Aerodynamic coefficients along the blade radius for *CaseT1.0* (solid), *CaseT1.2* (dashed), *CaseT1.4* (dashed dot) and *CaseT1.4-twistmod* (dashed dot dot).

459 the outer rotor sections as one might have expected due to a displacement effect of the nacelle. In total, the torque generated
 460 by one blade decreased for *CaseT1.4* by 1.18% compared to *CaseT1.0*.

461 4.4 Movement of the Blade Position Relative to the Nacelle

462 As pointed out in section 4.3.1, a segregation of the interacting pressure gradients of the blade and the nacelle could reduce the
 463 overall loading on the corner boundary layer. In order to analyze these effects, the blade was shifted upstream in axial direction
 464 (*CaseT1.4-dXm04*), as well as upstream- and downstream in circumferential direction (*CaseT1.4-dYm05* and *CaseT1.4-dYp05*,
 465 respectively). All modifications were based on the nacelle with the largest of the considered thicknesses, since first, the strongest
 466 effects on the flow separation might be present there, and further, this configuration provides more space for relative blade
 467 shifts. The motivation for the first modification was to place the entire blade in front of the point of maximum thickness of
 468 the nacelle and, therefore, locate it in the favorable pressure gradient in axial direction. The shifting of the blade in y -direction
 469 correspondingly aims to investigate the relative location of the pressure gradients in circumferential direction. Bangga (2018)

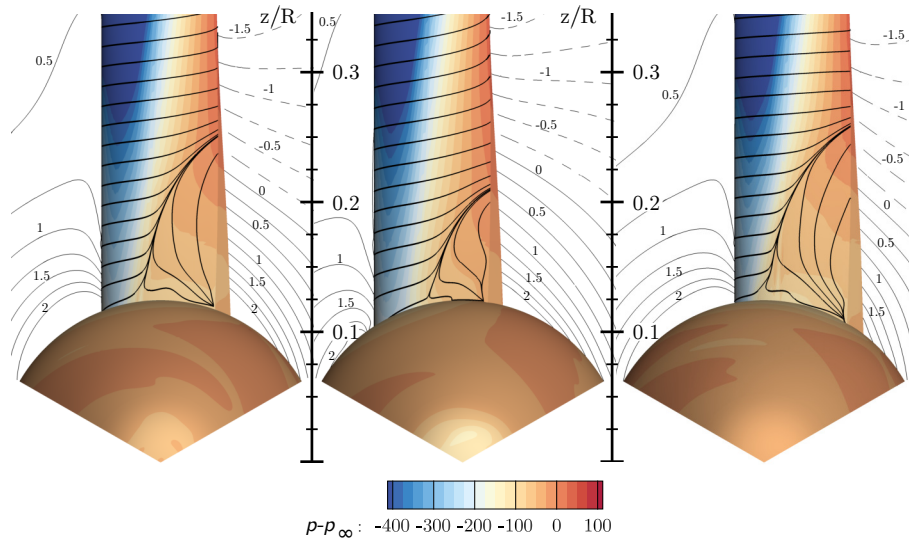


Figure 23. Visualization of three dimensional flow in the root region using streamlines and pressure contours for *CaseT1.4-dXm04* (left), *CaseT1.4-dYm05* (mid) and *CaseT1.4-dYp05* (right). The contour lines are plotted for a slice at $x = 2\text{m}$ ($x = 1.5\text{m}$ for *CaseT1.4-dXm04*) and indicate the velocity $\tilde{w} = w - \Omega y$ which denotes the deviation from the ideal kinematic path.

470 investigated the lateral shift of rotating profiles for the first time and focused on the exploitation of the y -component of the
 471 centrifugal force in Eq. 1 to increase rotor performance.

472 4.4.1 The Effect of the Relative Blade Position on Flow Separation

473 Comparing the surface streamlines plotted in Fig. 23 with those of the centered version (Fig. 16), the movement of the blade
 474 forward in axial direction reveals no improvement regarding the extension of the corner separation. Indeed, even a slight
 475 deterioration is present, which might be caused by the fact that the inclination angle relative to the blade increases. Turning to
 476 *CaseT1.4-dYp05*, an increase of the separation extent by around 8% can be observed compared to *CaseT1.4*. Here, separation
 477 already begins close behind 30% of chord. A significant improvement can be achieved by shifting the blade in direction of
 478 rotation. The separation size in radial direction is $0.086R$, which is only slightly above the baseline *CaseT1.0* but already
 479 smaller than in *CaseT1.2*.

480 The boundary layer profiles in the reference cut $r/R = 0.136$ (Fig.24) give information about the mass flow rate of the
 481 recirculation. At $x_c/c = 0.5$, *CaseT1.4-dYm05* is still attached, whereas slight and moderate back flow is observed in *CaseT1.4*
 482 and *CaseT1.4-dYp05*, respectively. Turning to the profiles at $x_c/c = 0.75$, the height of separation massively increases when
 483 shifting the blade in positive y -direction. As seen in Fig. 23 this seems to be linked with the volume covered by the downward
 484 curvature after the point of maximum thickness of the nacelle and might be "felt" by the blade boundary layer as an additional
 485 expansion, which goes along with added adverse pressure loading. This effect is weaker for *CaseT1.4-dYm05*, where most of
 486 the pressure recovery is achieved forward of that point.

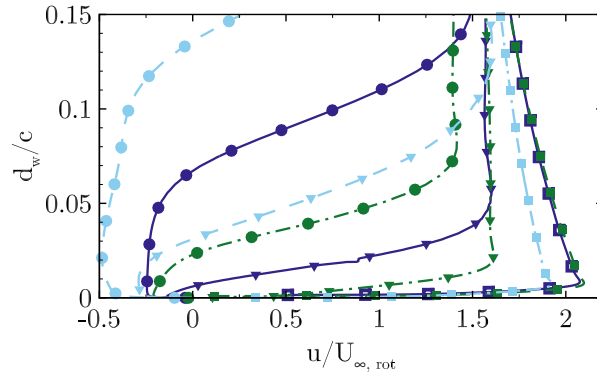


Figure 24. Stream-wise normalized velocity in the boundary layer $u/U_{\infty, rot}$ at $r/R = 0.136$ and $x_c/c = [0.25; 0.5; 0.75]$ (squares; triangles; circles), for *CaseT1.4* (solid), *CaseT1.4-dYm05* (dashed dot) and *CaseT1.4-dYp05* (dashed).

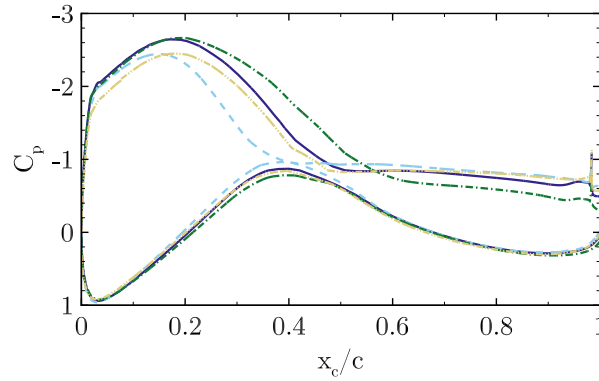


Figure 25. Airfoil pressure distribution at $r/R = 0.136$ for *CaseT1.4* (solid) *CaseT1.4-dXm04* (dashed dot dot), *CaseT1.4-dYm05* (dashed dot) and *CaseT1.4-dYp05* (dashed).

487 In a similar way, the chord-wise (Fig. 25) pressure distributions are affected in accordance to the previous observations. In
 488 *CaseT1.4-dXm04*, the suction peak decreases, but the kink to the pressure plateau remains at the same position as for *CaseT1.4*.
 489 In the aft-chord region the pressure recovery has still not initiated as it might be "blocked" by the displacement effect of the
 490 ascending nacelle diameter in the vicinity of the blade suction side. In *CaseT1.4-dYp05*, the massive flow separation yields a
 491 collapse of lift at $x_c/c \approx 0.4$, which results in a reduced suction peak. Both cause a tremendous increase of pressure drag. The
 492 latter can certainly be reduced for *CaseT1.4-dYm05*, where moderate separation begins at $x_c/c \approx 0.65$. The downward slope
 493 shows that the last part of the pressure recovery already occurs along the airfoil and not predominantly in the wake, as for the
 494 other cases.

495 Regarding the span-wise pressure distribution (Fig. 26), a slight increase of suction force can be noted near the leading edge
 496 in *CaseT1.4-dYm05* for the whole inner portion, whereas it is markedly lower when moving the blade in the other direction.
 497 In contrast to the other cases, *CaseT1.4-dYm05* maintains the slope of the leading edge cut, also along $x_c/c = 0.5$. Particularly,

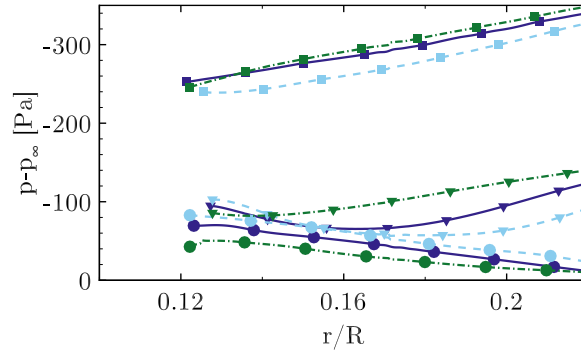


Figure 26. The effect of the blade position on the radial pressure distribution on the suction side, at $x_c/c \approx 0.1$ (squares), $x_c/c \approx 0.5$ (triangles) and $x_c/c \approx 0.85$ (circles) for *CaseT1.4* (solid), *CaseT1.4-dYp05* (dashed), *CaseT1.4-dYm05* (dashed dot).

498 *CaseT1.4-dYp05* already shows the behavior typically found near the trailing edge, where pressure recovery is redeployed in
 499 radial direction by the centrifugal pumping mechanism. Consistently at $x_c/c = 0.85$, the radial pressure gradient is smallest in
 500 *CaseT1.4-dYm05*, since less recirculating mass needs to be transported outward, which directly yields an earlier realignment
 501 of the streamlines in chord-wise direction as seen in Fig. 23.

502 From the previous investigations in Sec. 4.3.1 using the Euler simulations, it became clear, that a decoupling of the ad-
 503 verse pressure gradients seems necessary to relief the boundary layer in the junction. As could be shown the axial shifting
 504 did not yield any improvement, so that it can be deduced that the predominant pressure gradient is the one in lateral direc-
 505 tion. This seems also reasonable as the flow in the junction is aligned with that direction. As the adverse pressure gradient
 506 imposed by the nacelle initiates at the outmost point in circumferential direction, it follows that the entire pressure recovery
 507 is additionally loaded in *CaseT1.4-dY05*. Opposed to that, the segregation of the adverse pressure gradients seems to work for
 508 *CaseT1.4-dYm05*.

509 Another important aspect to be considered is already attributed to the inflow. As shown by the left sketch in Fig. 27, the
 510 inner sections of the blade which is shifted forward are affected by inflow stemming from an effectively larger radius that
 511 consequently brings in greater momentum. Along the airfoil the flow is pushed downward compared to its ideal kinematic path,
 512 which means that it is effectively streaming from larger to smaller radius. According to conservation of angular momentum
 513 this results in acceleration, which is supportive to overcome the adverse pressure gradient. When crossing the xz -plane a
 514 corresponding retardation would prevail, which additionally reduces angular velocity in *CaseT1.4-dYp05*. To illustrate this
 515 inflow hypothesis, a velocity difference plot between *CaseT1.4-dYm05* and *CaseT1.4* is presented in Fig. 27 for the reference
 516 cut $r/R = 0.136$. For this visualization, the solution of *CaseT1.4-dYm05* was mapped on top of the centered version. For the
 517 entire front part of the airfoil a higher velocity magnitude prevails, which is particularly present inside the boundary layer
 518 as seen before in Fig. 24. In the stagnation region the inflow velocity is $\approx 1\text{m/s}$ higher for *CaseT1.4-dYm05* compared to
 519 *CaseT1.4*. Another distinct peak, where the velocity is markedly higher for *CaseT1.4-dYm05* is found in the region of adverse

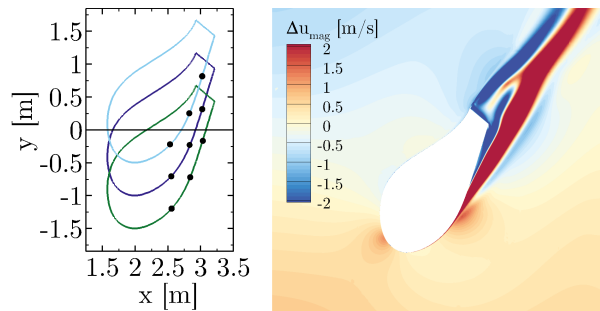


Figure 27. Sketch of the shifted profiles at $r/R = 0.136$ (left). Difference in velocity magnitude between *CaseT1.4-dYm05* and *CaseT1.4* (right).

520 pressure at around $x_c/c \approx 0.5$. In the rear part of the airfoil the differences originate from the different thickness of the separated
 521 wake.

522 The third effect that is seen to be beneficial for the delay of separation by shifting the blade in direction of rotation is
 523 the sweep angle of the inflow vector with respect to the blade leading edge. At the reference cut it comprises more than
 524 25° for *CaseT1.4-dYm05*, about 17° for *CaseT1.4* and around 9° for *CaseT1.4-dYp05*. In radial direction the sweep angle
 525 decreases exponentially, but is still greater than five degrees at the tip for *CaseT1.4-dYm05*. As the flow turns about the airfoil,
 526 the effective sweep angle is not constant along the chord-wise direction, but since most of the lift is generated in the front
 527 portion of the blade, it can certainly be stated that sweep effects cannot be neglected. This is supported by the streamlines in
 528 *CaseT1.4-dYm05* (Fig. 23) that show the typical pattern found for swept wings at high AoA (Obert, 2009), featuring a distinct
 529 attachment line instead of a stagnation point, which results in $C_{p,max} < 1$, as well as a prominent span-wise deflection of the
 530 streamlines near the trailing edge. In contrast to that, the streamlines remain approximately perpendicular to the leading edge
 531 in *CaseT1.4-dYp05*. The question, whether the principle of independence holds or not is certainly debatable, since rotation
 532 introduces span-wise gradients to the flow field. Indeed, for attached boundary layers it is a typical assumption made in BEM
 533 codes and is also supported by analyses of Leishman (1989). At high AoA in the stall regime it is generally not valid anymore.
 534 However, as observed in the boundary layer profiles (Fig. 24), the streamlines (Fig. 23) and pressure distribution (Fig. 25),
 535 there is evidence that the sweep angle delays stall and is beneficial for reattachment as it stimulates the outward centrifugal
 536 pumping mechanism. An overview of the effect of sweep angle on dynamic stall can be found in the text book of Leishman
 537 (2006). Measurements reinforcing the present observations regarding the effect of sweep angle on maximum lift coefficient
 538 and stall delay can be found for example in Dwyer and Aiccroskey (1971), or in Purser and Spearman (1951).

539 4.4.2 The Effect of the Blade Position Relative to the Nacelle on Aerodynamic Coefficients

540 **R2:G4-c** Regarding the global consequences of the relative blade position, those shall be compared in terms of AoA, lift- and
 541 drag coefficients plotted for the inner rotor half in Fig. 28. The AoA seems generally to increase when flow separation becomes
 542 stronger which was already seen for the cases where the nacelle thickness had been increased. Compared to *CaseT1.4*, the axial

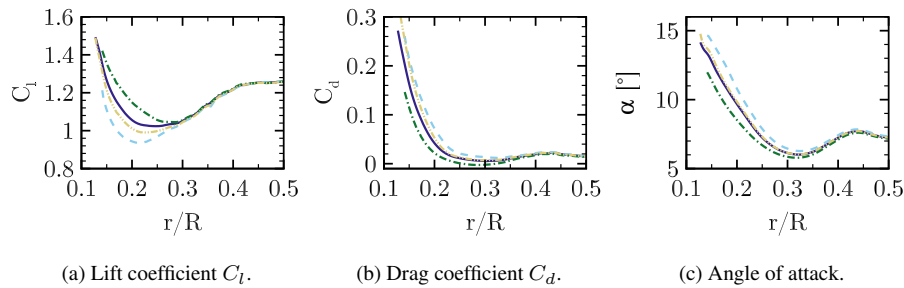


Figure 28. Aerodynamic properties along the blade radius for *CaseT1.4* (solid), *CaseT1.4-dXm04* (dashed dot dot), *CaseT1.4-dYm05* (dashed dot) and *CaseT1.4-dYp05* (dashed).

543 movement of the blade shows to decrease torque by 0.96% as lift declines and drag slightly rises. The significantly stronger
 544 flow separation in *CaseT1.4dYp05* further deteriorates aerodynamic efficiency and is reflected in a decline of torque by even
 545 1.98%. *CaseT1.4dYm05* clearly increases lift and decreases drag in the inner portion of the rotor which raises torque by 2.1%.
 546 This configuration is already better than *CaseT1.2*, but still around 1.5% worse than *CaseT1.0*. However, as will be shown in
 547 the next section, a movement of the blade in direction of rotation based on *CaseT1.0* is able to outperform the aerodynamic
 548 behavior of the latter.

549 4.5 Fillet-Type Modifications in the Junction

550 The second strategy for a potential reduction of corner separation elaborated in Sec. 4.3.1 was the relief of the boundary layer
 551 interaction of blade and nacelle which shall be addressed in this section. The considered configurations modify the geometry of
 552 the junction by applying a constant radius of 0.4m all around the airfoil, denoted *CaseT1.0-rounded* and introducing a fairing on
 553 the suction side of the blade referred to as *CaseT1.0-fairing*. These modifications are based on *CaseT1.0*. Since the movement
 554 of the blade in direction of rotation showed promising results when being based on the largest nacelle, this modification shall
 555 be transferred to the smallest nacelle (*CaseT1.0-dYm05*) and be included for comparison in the present discussion. In the rear
 556 part the junction reveals a plateau which, from another case not shown here (Kühn et al., 2018), turned out to be beneficial
 557 regarding a restriction of the recirculation area. Since the lateral shift on this nacelle brings along conflicts regarding meshing
 558 when moving the blade forward, the periodic segment had to be rotated by 17° , but due to axisymmetry, this does not change
 559 anything regarding periodicity.

560 4.5.1 The Effect on Flow Separation and the Corner Vortex System

561 The streamline plot in Fig. 29 shows that flow separation can be entirely suppressed for *CaseT1.0-dYm05* and *CaseT1.0-fairing*.
 562 For the latter quasi two-dimensional flow conditions prevail with only slight streamline deflection close to the trailing edge,
 563 whereas the shifted blade again shows the streamline patterns of swept wings. In *CaseT1.0-rounded*, flow separation cannot be
 564 noticeably reduced compared to *CaseT1.0* (Fig. 9).

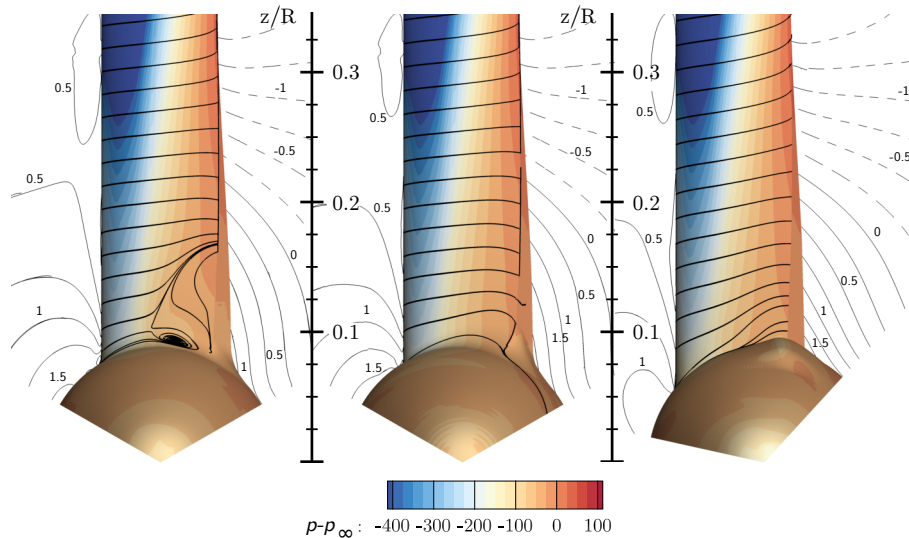


Figure 29. Visualization of three dimensional flow in the root region using streamlines and pressure contours for *CaseT1.0-rounded* (left), *CaseT1.0-fairing* (mid) and *CaseT1.0-dYm05* (right). The contour lines indicate the "deviation velocity" \tilde{w} from to the ideal kinematic value $-\Omega y$ in a slice at $x = 2m$.

565 More details on the separation mechanism (and suppression) can be gathered from the vortex system in the corner shown
 566 in Fig. 30. Although the footprint of the surface streamlines indicate a similar separation size for *CaseT1.0-rounded* compared
 567 to the baseline configuration with sharp juncture, recapitulation of Fig. 20 confirms that the separation thickness decreased by
 568 introducing the rounding. A side effect of the latter is a weakening of the production of the HSV in the nose region, achieved by
 569 a homogenization of the shear layer shown in the foremost slice of the approaching nacelle boundary layer interacting with the
 570 blade. Although a streamline which is counter-rotating to the HSV vortex rotating can be identified in its vicinity, no harmful
 571 corner vortex can build up and manifest itself in the junction. Separation is not initiated from a distinct streamline, but evolves
 572 from an isolated vortex generated in the transition from the rounding to the actual blade surface.

573 These findings were beared in mind in the design of *CaseT1.0-fairing*. It was decided to reduce the fillet nose radius to 0.2m,
 574 in order to increase the strength of the HSV again, which is believed to be helpful for reduction of corner separation (Devenport
 575 et al., 1992). This is certainly the conservative approach, since the separation should optimally be suppressed in combination
 576 with a reduced or even eliminated HSV, as the latter increases the interference drag and is a source of noise (Zess and Thole,
 577 2001), (Simpson, 2001), (Devenport et al., 1992). Towards the trailing edge, the local radius was increased on the suction side to
 578 eliminate the unfavorable, pressure rise induced by the shape in the transition to the blade previously seen in *CaseT1.0-rounded*.
 579 As can be observed in the streamlines as well as in the foremost slice plotting the vorticity contours, the strength of the HSV
 580 could be increased by a factor of three. Its suction side leg remains closer to the blade, as it is not displaced by any recirculation
 581 and is deformed ovally when traveling downstream.

582 In the work of Bordji et al. (2015), the formation of the corner vortex correlated well with a peak in the Reynolds shear stress
 583 generated by the chord-wise and span-wise velocity. In order to assess the development of the corner vortex for *CaseT1.0* and
 584 *CaseT1.0-fairing*, the contours of the $\langle vw \rangle$ shear stress are depicted in Fig. 31 for a field slice cutting the blade at $x_c/c = 0.5$.
 585 The plot confirms the peak and the change in sign of shear stress for *CaseT1.0* similarly as observed in Bordji et al. (2015)
 586 and gives evidence for the production of the corner vortex. In *CaseT1.0-fairing*, the large rounding radius ensures a smooth
 587 mixing of the boundary layers resulting in a homogeneous distribution of shear stress which prevents the generation of the CV.
 588 Additionally, as it was already suggested by the streamline plot in Fig. 29, the formation of the HSV is altered, as well. Its
 589 strength is lower than in *CaseT1.0* and it is stretched. The fact that its center is located significantly closer to the blade surface
 is beneficial regarding its induction on the blade.

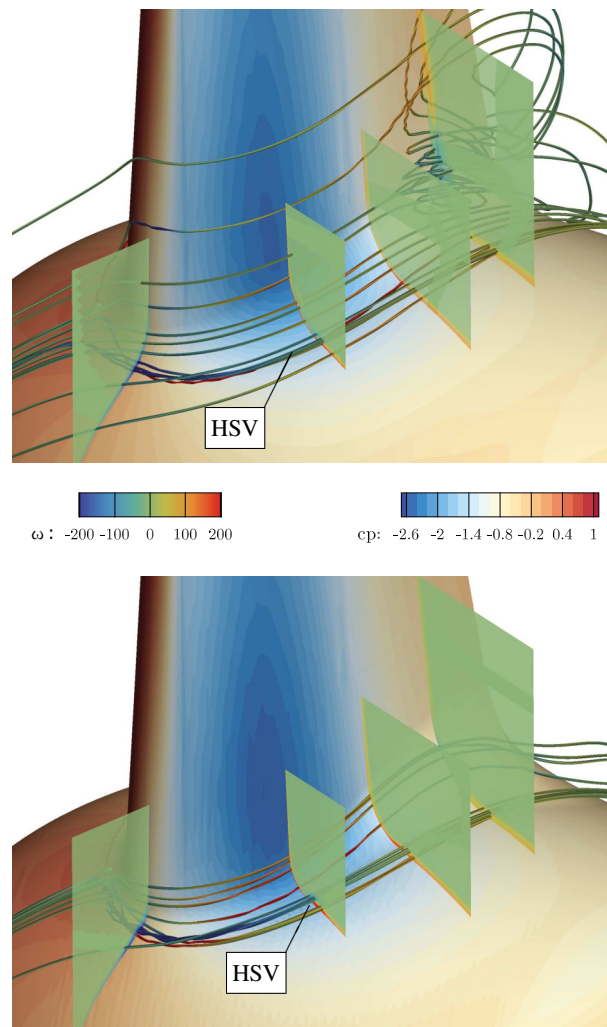


Figure 30. Vortex system in the junction of blade and nacelle for *CaseT1.0-rounded* (upper) and *CaseT1.0-fairing* (lower).

590

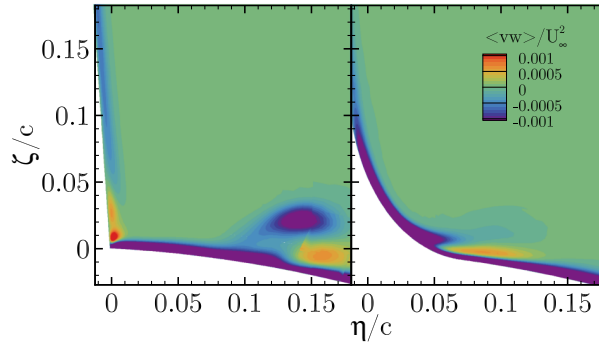


Figure 31. Reynolds shear stress $\langle vw \rangle$ of the chord-wise and span-wise velocity at $x_c/c = 0.5$. *CaseT1.0* (left), *CaseT1.0-fairing* (right).

591 4.5.2 The Effect of Fillets on the Aerodynamic Coefficients

592 Turning finally to the quantities related to aerodynamic efficiency, the pressure distribution in Fig. 32 clearly shows the im-
 593 provement obtained by eliminating the corner separation for *CaseT1.0-dYm05* and *CaseT1.0-fairing*. The suction force is
 594 particularly larger between 40-60% chord and the boundary layer is able to stand the adverse pressure gradient which leads to
 595 an effectively higher pressure value in the vicinity of the trailing edge.

596 The attached flow in the corner reduces the AoA (Fig. 33 c)), consistently to the previous observations, being most pro-
 597 nounced for *CaseT1.0-dYm05*. As already indicated by the pressure distribution in Fig. 32, *CaseT1.0-fairing* particularly in-
 598 creases lift and only marginally decreases drag, whereas for *CaseT1.0-dYm05* the improvements are more related to drag.
 599 Overall, tangential force increases in *CaseT1.0-fairing* due to greater lift. The evaluation of these local improvements on the
 600 global blade performance is discussed in the next section.

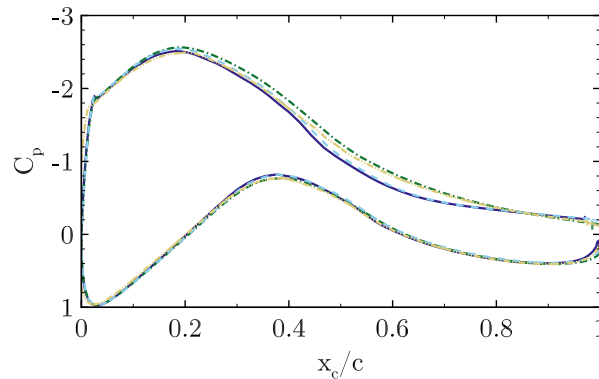


Figure 32. Pressure coefficient at $r/R = 0.136$. *CaseT1.0* (solid), *CaseT1.0-rounded* (dashed), *CaseT1.0-fairing* (dashed-dot), *CaseT1.0-dYm05* (dashed dot dot).

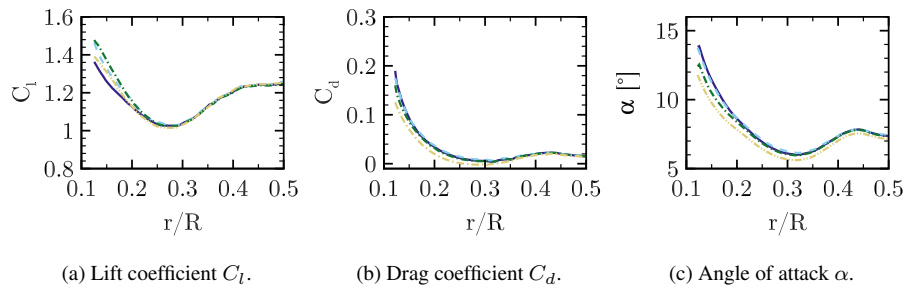


Figure 33. Distribution of aerodynamic coefficients along the blade radius for *CaseT1.0* (solid), *CaseT1.0-rounded* (dashed) *CaseT1.0-fairing* (dashed dot) *CaseT1.0-dYm05* (dashed dot dot).

601 4.6 Assessment of Integral Quantities and Off-Design Conditions.

602 **R1:G6-b** **R2:G4-d** As previously mentioned, it is important to put the very local aerodynamic improvements in a more
 603 global context, by analyzing their impact on the sectional thrust and driving loads of the entire blade and by comparing the
 604 associated integral forces and moments. It is further important to evaluate the performance in off-design conditions. For this
 605 reason the fairing modification, which turned out to be the most promising candidate to eliminate flow separation in the junc-
 606 tion of the blade and the nacelle, is investigated in comparison to the baseline geometry for further wind speeds, namely
 607 $U_\infty = 8; 12; 15$ m/s. Those correspond to the tip-speed ratios of 7.48, 5 and 4, respectively. Since the pitch angle is kept con-
 608 stant, the resulting pitch misalignments render somewhat off-design "atmospheric" conditions. The two main questions to be
 609 answered are whether the modified fairing geometry wastes performance at lower than the design wind speed and whether an
 610 additional benefit can be obtained at high wind speeds, where the overall tendency of flow separation increases. The latter fact
 611 can be important with respect to atmospheric turbulence, since gusts may cause local flow separation. If the amount of flow
 612 separation can be reduced, an overall reduction of load fluctuations could be achieved.

613 For the reason that unsteadiness is expected to increase with wind speed, the computations at 12 and 15 m/s were continued
 614 unsteady from a steady-state solution for two more revolutions. Time-averaging of the output data was conducted for the last
 615 revolution. An overview on the effects of the wind speed on the flow field can be gained from Fig. 34, where a comparison is
 616 drawn between *CaseT1.0-fairing* and *CaseT1.0* based on the surface distributions of pressure and streamlines, as well as based
 617 on the chord-wise velocity in an airfoil section at $z/R = 0.2$. At the lowest wind speed $U_\infty = 8$ m/s corner separation has
 618 also almost vanished for the baseline geometry and very similar pressure, streamline and velocity distributions are obtained
 619 in both cases. For higher wind speeds the AoA increases, yielding a stronger acceleration of the flow in the front part of the
 620 airfoil and accordingly in very low surface pressures. As expected, the area of separation increases in chord-wise and radial
 621 direction. After detachment of the flow, the pressure contours flatten out in conjunction with a strong effective de-cambering of
 622 the airfoils. This is reflected in a significant recirculation area shown in the velocity contours. Overall, *CaseT1.0-fairing* signif-
 623 icantly reduces the separated area for the high wind speed cases. Particularly, the thickness of the separated wake is markedly
 624 reduced and the accelerated regime in the front part of the airfoil is maintained. Both result in a redirection of the airfoil wake,

625 which consequently turns also of the aerodynamic force vector. The resulting sectional thrust and driving loads of the blade
626 are shown in the Figs. 35 and 36. Their integrated values resulting rotor thrust, driving force and torque (for one blade) are
627 summarized in Tab. 1. It must be pointed out that for the lower wind speeds ($U_\infty = 8; 10 \text{ m/s}$) an independent integration of
628 each entire curve was not possible, since the effect of the fairing on the overall performance is very small. The latter fact is
629 not surprising when keeping in mind that the flatback blade offers only little room for improvement compared to the massively
630 separated root sections of conventional blades. The problem is that particularly for torque, the smallest numerical uncertainties
631 in the prediction of the forces in the outer part of the rotor can outweigh the small improvements obtained in the inner sections.
632 Although, the results shown in the Figs. 5 and 6 imply very small influence of the grid on the solution, it must be remembered
633 from section 3.2 that the two cases compared here cannot use exactly the same grids due topological reasons. Therefore, influ-
634 ences of the grid cannot be eliminated completely. Nevertheless, to be able to examine the influence of the inner portion on the
635 overall performance, each curve was integrated separately only up to $r/R \leq 0.35$. The result was then superposed in both cases
636 by the same value calculated from the baseline configuration for $r/R > 0.35$. For the high wind speed cases $U_\infty = 12; 15 \text{ m/s}$
637 each curve was independently integrated over the whole radius. At the lowest wind speed $U_\infty = 8 \text{ m/s}$ the phenomenological
638 impression made before is confirmed in the forces and integral quantities, which are almost identical in both cases. Hence,
639 the first question, whether a performance degradation of the fairing is present at low wind speeds can be negated. For the
640 design wind speed $U_\infty = 10 \text{ m/s}$ the local improvement of the aerodynamics discussed in the previous section is estimated to
641 evoke a very small overall improvement of the extracted power by around 0.1%. When increasing the wind speed, the relative
642 improvements compared to the baseline configuration become more pronounced. In particular, a rise of the driving force can
643 be observed, which can be related to the relative decrease of the pressure drag caused by the separated airfoil wake. For the
644 largest wind speed this means an overall power gain of 0.54%.

645 To summarize these findings it can be stated that it is still important to reduce the stall tendency of wind turbine blades in
646 the inboard sections, even if it might be not a severe problem at the design point of operation, since at higher wind speeds and
647 pitch misalignment flow separation will always expand from the root in outward direction. Therefore, the smaller the triggering
648 flow separation near the root is, the less is the mass of separated flow that has to be transported outwards by the centrifugal
649 pumping mechanism, which means that "clean" aerodynamic behavior can be maintained in the outer part of the rotor. With
650 respect to situations of very high atmospheric turbulence levels, where such temporal pitch misalignments cannot be avoided,
651 an overall performance improvement can be expected in conjunction with a reduction of load fluctuations.

652 5 Conclusions

653 Numerical investigations on the flow in the root region of a wind turbine with flatback airfoils have been performed using the
654 CFD code FLOWer.

655 The flow in the blade root region has been characterized and the influence of the nacelle on the root flow has been elaborated
656 by comparison with an isolated rotor simulation. Flow separation in the corner of the blade and the nacelle has been identified
657 as one effect that deteriorates the aerodynamic efficiency there. This flow feature is different to the commonly known massive

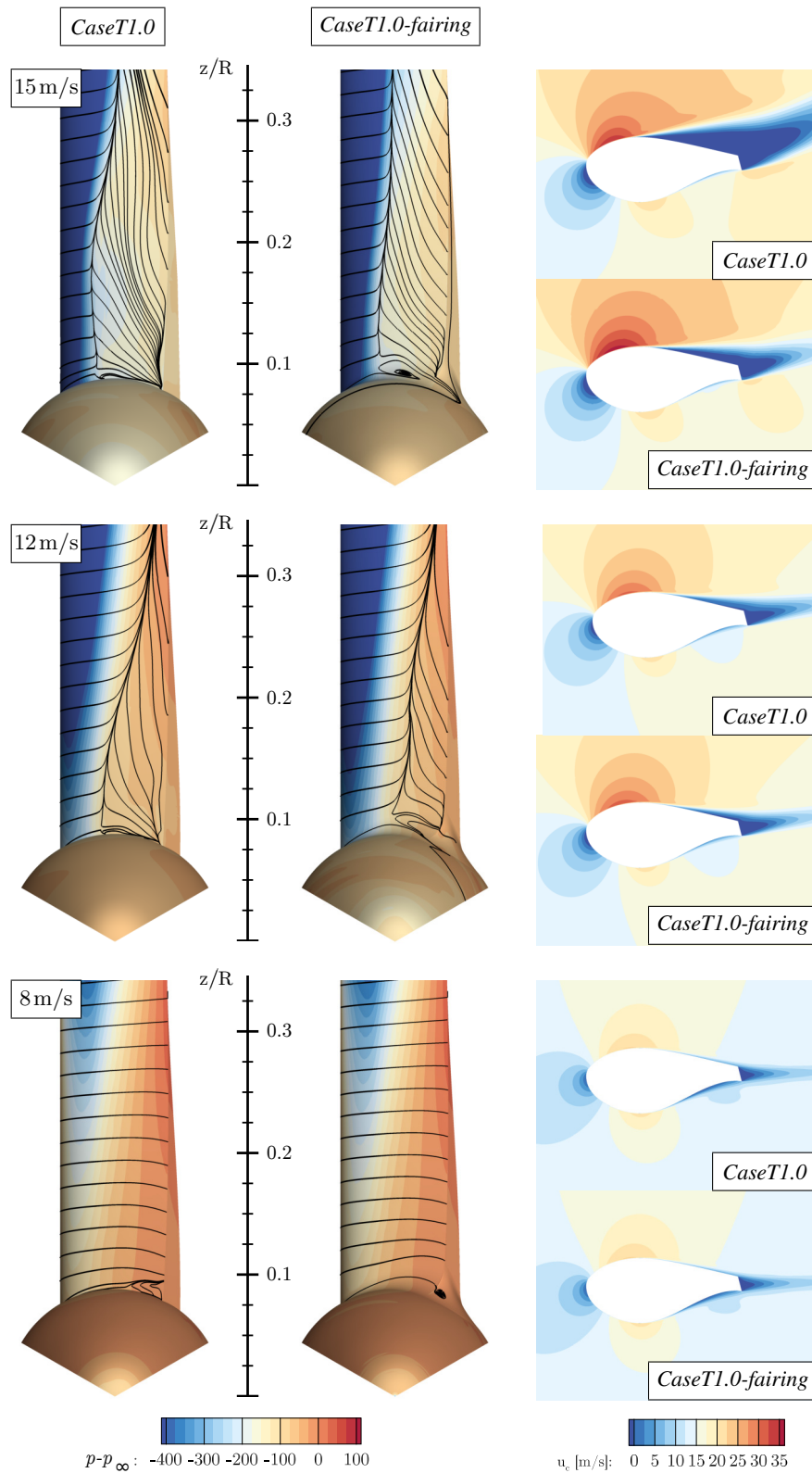


Figure 34. Comparison of the flow field in the root region for *CaseT1.0* and *CaseT1.0-fairing* at off-design wind speeds: $U_\infty = 8; 12; 15\text{m/s}$. Surface streamlines (left) and chord-wise velocity distribution at $z/R = 0.2$ (right).

Table 1. Integral blade forces and torque for *CaseT1.0* and *CaseT1.0-fairing* at different wind speeds.

Thrust force [kN]		
Wind speed [m/s]	<i>CaseT1.0</i>	<i>CaseT1.0-fairing</i>
8	16.442	16.448 (+0.040%)
10	23.187	23.212 (+0.108%)
12	28.721	28.850 (+0.448%)
15	32.878	33.113 (+0.714%)
Driving force [kN]		
Wind speed [m/s]	<i>CaseT1.0</i>	<i>CaseT1.0-fairing</i>
8	1.624	1.628 (+0.202%)
10	4.128	4.142 (+0.351%)
12	6.936	7.029 (+1.336%)
15	10.637	10.838 (+1.891%)
Torque [kNm]		
Wind speed [m/s]	<i>CaseT1.0</i>	<i>CaseT1.0-fairing</i>
8	20.309	20.326 (+0.082%)
10	50.873	50.923 (+0.099%)
12	84.387	84.833 (+0.528%)
15	127.007	127.693 (+0.540%)

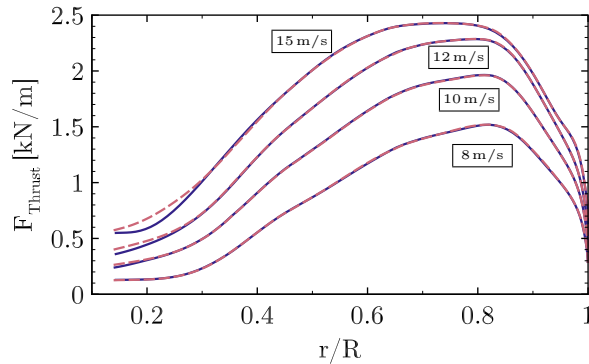


Figure 35. Influence of different wind speeds on the sectional thrust force. *CaseT1.0* (solid), *CaseT1.0-fairing* (dashed).

658 flow separation in the root region of conventional turbines with cylindrical root sections, as it is governed by a vortex system
 659 evolving in the junction of the blade and the nacelle. Although the separation is shallow, the very high solidity of the rotor

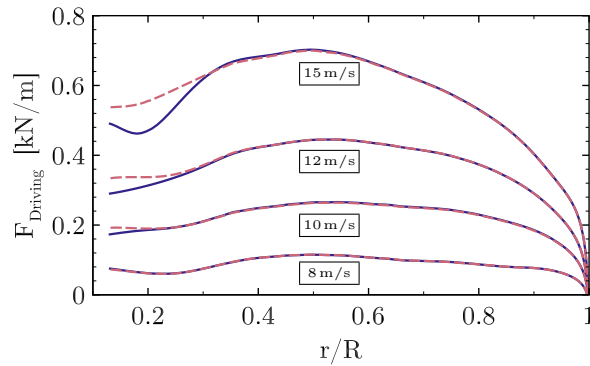


Figure 36. Influence of different wind speeds on the sectional driving force. *CaseT1.0* (solid), *CaseT1.0-fairing* (dashed).

660 involves strong three-dimensional effects which are determined by centrifugal and Coriolis forces as well as significant sweep
 661 angles. It can be stated that when focusing on blade root aerodynamics for those kind of rotors, a consideration of the nacelle
 662 seems to be highly important, as flow topology, aerodynamic coefficients and forces were markedly different when simulating
 663 only the isolated rotor.

664 Following the principal analyses, geometrical properties affecting the blade-nacelle interference have been introduced with
 665 the objective to better understand the underlying flow physics and eventually improve the aerodynamic performance of the
 666 rotor in the root region. The first parameter investigated was the relative nacelle thickness. By increasing the nacelle thickness,
 667 flow separation in the root region drastically increased, resulting in power losses for the whole rotor of up to 1.18%. The
 668 reasons for the increased flow separation were found from inviscid reference simulations indicating a higher overall adverse
 669 pressure gradient for the larger nacelle thickness by means of an unfavorable interaction of the pressure minimums of the blade
 670 and the nacelle. An important observation of the viscous simulations was also that the increased nacelle thickness intensifies
 671 the detrimental interaction of the boundary layers in the junction area. From these findings two threads for diminishing the
 672 separation have been elaborated, the first was a decoupling of the pressure gradients of the blade and the nacelle and the second
 673 aimed for an alleviation of the harmful boundary layer interaction in the junction region.

674 A shift of the blade forward in axial direction did not improve the stall behavior. More effective was the lateral decoupling of
 675 the pressure gradients. By moving the blade forward in lateral direction, stall could be effectively reduced, or even eliminated
 676 when applying it to the smallest nacelle thickness, and with that, aerodynamic efficiency could be increased. The mechanisms
 677 for the improvements were explained by the decoupling of the pressure gradients of the blade and the nacelle, and by kinematic
 678 reasons which resulted in higher velocities in the front part of the blade and in the region of the adverse pressure gradient. An
 679 effective sweep angle further supported the outward transport of separated flow. The shift of the blade in the other direction
 680 deteriorated the flow dynamics in the root region compared to the centered base version.

681 The attachment of fillets in the junction of the blade and the nacelle could be shown to diminish flow separation, as well.
 682 Particularly effective was the installation of a fairing, implying a small rounding radius at leading edge and a large radius near
 683 the trailing edge of the suction side. With that configuration, separation could be completely eliminated.

684 **R1:G6-c** **R2:G4-e** The comparison with the baseline geometry showed that for the design wind speed the improved
685 aerodynamic behavior in the inner part of the rotor yields only a small improvement of the overall rotor torque. For off-design
686 conditions it could be shown that a decrease in wind speed yields that flow separation almost disappears for the baseline geom-
687 etry, as well. The fairing modification does not diminish the performance in that case. For higher wind speeds, flow separation
688 becomes generally more severe and the relative improvements obtained by installing the fairing are more pronounced. For
689 the latter, the area of separated wake in the very inboard region of the rotor is significantly reduced which improves whole
690 aerodynamics for the inner one third of the blade and enhances rotor performance in the order of 0.5%. Hence, as an outlook,
691 an overall performance gain and a decrease of load fluctuations can be expected for turbulent inflow conditions with high levels
692 of turbulence intensity. Investigations on that are dedicated to future work.

693 Apart from such analyses, it must be stated that this study was meant to demonstrate basic parameters affecting the root flow
694 of a wind turbine with aerodynamically shaped sections down to the hub. Potential for improvement could certainly lie in a
695 detailed parameter optimization of any of the investigated branches. For example it could be aimed to design fillets that prevent
696 separation and effectively alleviate the horse shoe vortex in order to decrease interference drag.

697 **Appendix A: The Effect of the Turbulence Model on Corner Separation**

698 The authors are aware that the prediction of corner flow separation can be rather sensitive to the choice of the turbulence
699 model (Vassberg et al., 2008; Rumsey et al., 2016). For this reason, a priori simulations of the simplified wing-body junction test
700 case of Gand et al. (2015) have been performed. In principle, this is a wing mounted on a flat plate of a certain length operating
701 at an AoA of 12° . Three hierarchically different turbulence models have been tested; the one equation SA model (Spalart and
702 Allmaras, 1992), the two-equation $k-\omega$ -SST model (Menter, 1994) and the SSG- ω differential Reynolds stress model (Eisfeld,
703 2004). The results visualizing the corner separation are compared with experimental oil coating in Fig. A1. The SA-model
704 expectedly to literature (Vassberg et al., 2008; Bordji et al., 2015) greatly over-predicts the amount of separation, whereas
705 the SSG- ω model predicts hardly any separation. In turn, the SST-model provides reasonable results regarding the separation
706 size, with a pattern quite similar to the one obtained by Bordji et al. (2015) using the SA-QCR model (Spalart, 2000). The
707 SST-model further showed meaningful results for vortex properties (HSV and CV) and turbulent kinetic energy (Przewlocki,
708 2017). As the full RSM is computationally more expensive and further more delicate concerning robustness, it was decided to
709 stick with the SST-model for the considered turbine simulations.

710 **Appendix B: Assessment of the Effect of Boundary Layer Transition**

711 **R1:G5-b** The effect of boundary layer transition on the flow separation and the turbine loading is elaborated by analyzing the
712 baseline configuration *CaseT1.0* with the transitional $\gamma-Re_\theta$ - $k-\omega$ -SST model, and a coupling of the SST model to the e^N -enve-
713 lope method (e^N - $k-\omega$ -SST). The critical N -factor is set constant to $N = 9$, and the free stream turbulence intensity entering the
714 γ equation of the $\gamma-Re_\theta$ model is adjusted such that a similar N -factor is obtained when applying the relation of Mack (1984).

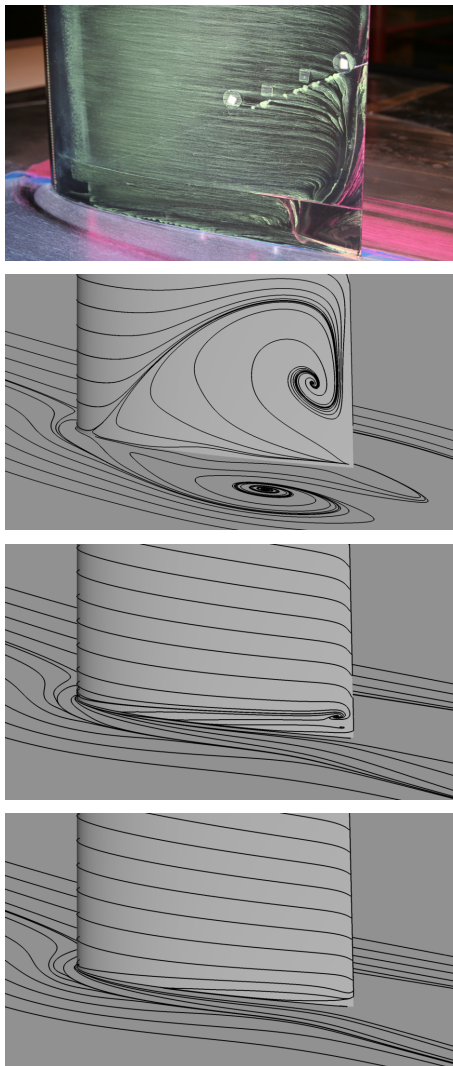


Figure A1. Visualization of corner separation for the simplified wing-body junction experiment of Gand et al. (2015) (upper most). Prediction with SA (upper), SST (mid), and RSM (lower). Figures reproduced from simulations of Przewlocki (2017).

715 Since laminar separation bubbles are expected, the transitional computations were conducted unsteady and the sectional loads
 716 are therefore presented time-averaged.

717 The transition location can be identified by the strong local rise of the wall shear stress which is plotted in Fig. B1 together
 718 with the surface streamlines for the fully turbulent and transitional simulations employing the $\gamma-Re_{\theta}$ model. In the front part
 719 of the nacelle the lower shear stress suggests laminar flow. At the inner section of the rotor, transition can be identified at
 720 $x_c/c \approx 0.2$. It should be noted that for the thick airfoils considered here, the arc length of laminar flow measured from the
 721 stagnation point is considerable. Hence, the "health" of the boundary layer with laminar history is very likely better compared

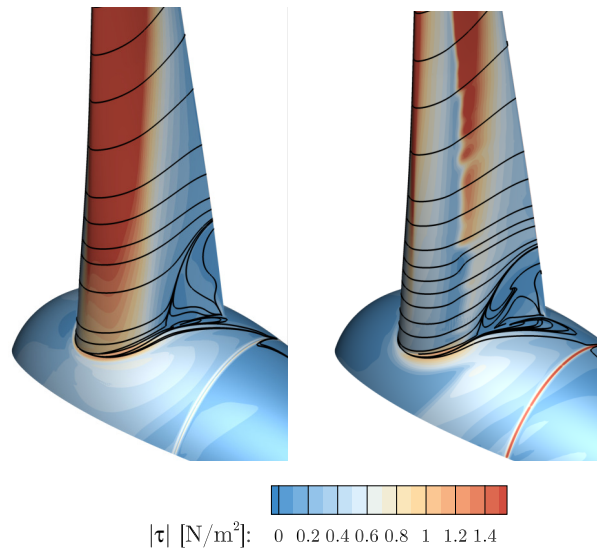


Figure B1. Surface streamlines and contours of shear stress magnitude for fully turbulent $k\text{-}\omega\text{-SST}$ simulation (left) and $\gamma\text{-}Re_\theta\text{-}k\text{-}\omega\text{-SST}$ simulation (right).

722 to the one which was assumed turbulent right from the beginning. When the boundary layer becomes subject to the adverse
 723 pressure gradient, an earlier separation can therefore be expected for fully turbulent conditions. Accordingly, it can be seen that
 724 the extent of the corner separation decreases when taking into account boundary layer transition. However, it can be also noted
 725 that the general flow topology does not change.

726 Turning to the sectional thrust and driving forces, cross-plotted in Fig. B2 and B3 for both, transitional and fully turbulent
 727 computations, at first glance, a good agreement of the $\gamma\text{-}Re_\theta$ and the e^N method is obtained over the whole span. For the thrust
 728 load, the difference between transitional and fully turbulent results are most evident in the outer rotor region. In that part, the
 729 turbulent boundary layer is very thick and already indicates very small separation near the trailing edge. When the flow has a
 730 laminar history no flow separation is visible there. The boundary layer is thinner which reduces the effective de-cambering of
 731 the airfoils so that they can generate more lift and have less drag. The latter increases the driving force. In the inner section of
 732 the rotor, the same mechanism holds which means increasing lift by alleviation of the stress to the boundary layer. Since the
 733 twist angles are large, the increasing lift is distinctively reflected in the driving force, as well.

734 It can be concluded that expectedly the accounting for boundary layer transition increases thrust and driving forces of the
 735 rotor, by increasing lift and reducing viscous drag. The span-wise and chord-wise extent of the corner separation is reduced,
 736 but no fundamental change of the topology is visible. As suggested by measurements such as those of Zamir (1981), transition
 737 in the corner region is typically earlier than for an equivalent flat plate boundary layer. Hence in reality, where also pollution
 738 and surface roughness plays a role, the extent of separation might lie in between the one predicted by the fully turbulent and the
 739 transitional simulation. Both facts justify the fully turbulent approach when comparing the different geometry modifications.

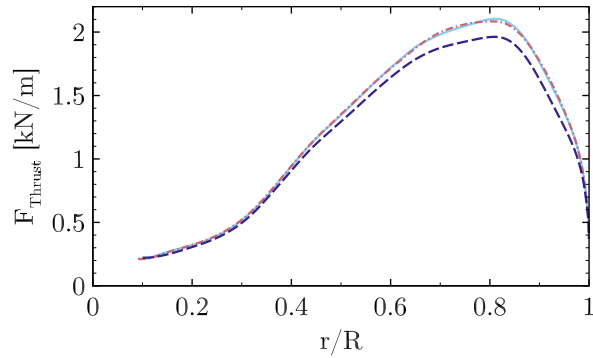


Figure B2. Influence of boundary layer transition on the sectional thrust force. Fully turbulent $k\text{-}\omega\text{-SST}$ (dashed), $\gamma\text{-}Re_\theta\text{-}k\text{-}\omega\text{-SST}$ (solid), $e^N\text{-}k\text{-}\omega\text{-SST}$ (dashed dot).

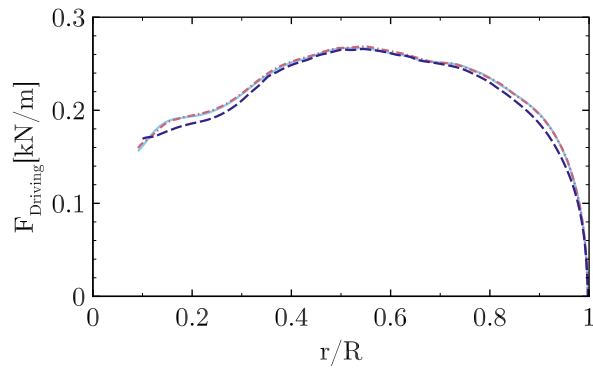


Figure B3. Influence of boundary layer transition on the sectional driving force. Fully turbulent $k\text{-}\omega\text{-SST}$ (dashed), $\gamma\text{-}Re_\theta\text{-}k\text{-}\omega\text{-SST}$ (solid), $e^N\text{-}k\text{-}\omega\text{-SST}$ (dashed dot).

740 Appendix C: Nomenclature

741 *Competing interests.* The authors declare that they have no conflict of interest.

742 *Acknowledgements.* The authors acknowledge the *German Federal Ministry for Economic Affairs and Energy* for funding this study as
 743 part of the project "AssiSt" (grant number 0325719A) and the *High Performance Computing Center Stuttgart* for the contribution of the
 744 computational resources.

745 References

- 746 Akay, B.: The root flow of horizontal axis wind turbine blades: Experimental analysis and numerical validation, 2016.
- 747 Arnal, D. and Juillen, J.: Three-dimensional transition studies at ONERA/CERT, in: 19th AIAA, Fluid Dynamics, Plasma Dynamics, and
748 Lasers Conference, p. 1335, 1987.
- 749 Bak, C., Zahle, F., Bitsche, R., Kim, T., Yde, A., Henriksen, L. C., Natarajan, A., and Hansen, M. H.: Description of the DTU 10 MW
750 Reference Wind Turbine, <http://dtu-10mw-rwt.vindenergi.dtu.dk>, 2017.
- 751 Baldacchino, D., Ferreira, C., Tavernier, D. D., Timmer, W., and van Bussel, G.: Experimental parameter study for passive vortex generators
752 on a 30% thick airfoil, *Wind Energy*, 2018.
- 753 Bangga, G.: Three-Dimensional Flow in the Root Region of Wind Turbine Rotors, kassel university press GmbH, 2018.
- 754 Bangga, G., Lutz, T., Jost, E., and Krämer, E.: Erratum:“CFD studies on rotational augmentation at the inboard sections of a 10 MW wind
755 turbine rotor”[*J. Renewable Sustainable Energy* 9, 023304 (2017)], *Journal of Renewable and Sustainable Energy*, 10, 019902, 2018.
- 756 Benek, J. A., Steger, J. L., Dougherty, F. C., and Buning, P. G.: in: *Chimera. A Grid-Embedding Technique.*, 1986.
- 757 Boorsma, K., Schepers, J., Gomez-Iradi, S., Herraez, I., Lutz, T., Wehling, P., Oggiano, L., Pirrung, G., Madsen, H., Shen, W., Rahimi, H.,
758 and Schaffarczyk, P.: Final report of IEA Task 29, Mexnet (Phase 3), IEA, 2018.
- 759 Bordji, M., Gand, F., Deck, S., and Brunet, V.: Investigation of a Nonlinear Reynolds-Averaged Navier–Stokes Closure for Corner Flows,
760 *AIAA Journal*, 2015.
- 761 Devenport, W. J., Simpson, R. L., Dewitz, M. B., and Agarwal, N. K.: Effects of a leading-edge fillet on the flow past an appendage-body
762 junction, *AIAA journal*, 30, 2177–2183, 1992.
- 763 Drela, M. and Giles, M. B.: Viscous-inviscid analysis of transonic and low Reynolds number airfoils, *AIAA journal*, 25, 1347–1355, 1987.
- 764 Du, Z. and Selig, M.: The effect of rotation on the boundary layer of a wind turbine blade, *Renewable Energy*, 20, 167–181, 2000.
- 765 Dwyer, H. and Aiccroskey, W.: Crossflow and unsteady boundary-layer effects on rotating blades, *AIAA Journal*, 9, 1498–1505, 1971.
- 766 Eisfeld, B.: Implementation of Reynolds stress models into the DLR-FLOWer code, 2004.
- 767 Gand, F., Monnier, J.-C., Deluc, J.-M., and Choffat, A.: Experimental study of the corner flow separation on a simplified junction, *AIAA*
768 *Journal*, 2015.
- 769 Grabe, C. and Krumbein, A.: Extension of the γ - $Re\theta_t$ model for prediction of crossflow transition, in: 52nd Aerospace Sciences Meeting, p.
770 1269, 2014.
- 771 Guntur, S. and Sørensen, N. N.: A study on rotational augmentation using CFD analysis of flow in the inboard region of the MEXICO rotor
772 blades, *Wind Energy*, 18, 745–756, 2015.
- 773 Herráez, I., Stoevesandt, B., and Peinke, J.: Insight into rotational effects on a wind turbine blade using Navier–Stokes computations, *Ener-*
774 *gies*, 7, 6798–6822, 2014.
- 775 Herráez, I., Akay, B., van Bussel, G. J., Peinke, J., and Stoevesandt, B.: Detailed analysis of the blade root flow of a horizontal axis wind
776 turbine, *Wind Energy Science*, 1, 89–100, 2016.
- 777 Himmelskamp, H.: Profile investigations on a rotating airscrew, MAP, 1947.
- 778 Jameson, A.: Time dependent calculations using multigrid, with applications to unsteady flows past airfoils and wings, *AIAA paper*, 1596,
779 1991, 1991.
- 780 Jameson, A., Schmidt, W., Turkel, E., et al.: Numerical solutions of the Euler equations by finite volume methods using Runge-Kutta time-
781 stepping schemes, *AIAA paper*, 1259, 1981, 1981.

782 Johansen, J. and Sørensen, N. N.: Aerofoil characteristics from 3D CFD rotor computations, *Wind Energy*, 7, 283–294, 2004.

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

785 Jost, E., Klein, L., Leiprand, H., Lutz, T., and Krämer, E.: Extracting the angle of attack on rotor blades from CFD simulations, *Wind Energy*.
786 Accepted, 2018.

787 Klein, L., Gude, J., Wenz, F., Lutz, T., and Krämer, E.: Advanced CFD-MBS coupling to assess low-frequency emissions from wind
788 turbines, *Wind Energy Science Discussions*, 2018, 1–30, <https://doi.org/10.5194/wes-2018-51>, <https://www.wind-energ-sci-discuss.net/wes-2018-51/>, 2018.

790 Knezevici, D., Sjolander, S., Praisner, T., Allen-Bradley, E., and Grover, E.: Measurements of secondary losses in a turbine cascade with the
791 implementation of nonaxisymmetric endwall contouring, *Journal of Turbomachinery*, 132, 011 013, 2010.

792 Kowarsch, U., Keßler, M., and Krämer, E.: High order CFD-simulation of the rotor-fuselage interaction, 2013.

793 Kroll, N., Rossow, C.-C., Becker, K., and Thiele, F.: The MEGAFLOW project, *Aerospace Science and Technology*, 4, 223–237, 2000.

794 Kühn, T., Altmikus, Daboul, H., Radi, A., Raasch, S., Knigge, C., Böske, L., Schwarz, T., Heister, C., Möller, A., Lutz, T., Weihing, P.,
795 Schulz, C., Thiemeier, J., Mockett, C., Fuchs, M., and Thiele, F.: AssiSt-Schlussbericht gemäß NKBF98, Tech. rep., 2018.

796 Langtry, R.: Extending the Gamma-Rethetat Correlation Based Transition Model for Crossflow Effects, in: 45th AIAA Fluid Dynamics
797 Conference, p. 2474, 2015.

798 Langtry, R. B. and Menter, F. R.: Correlation-based transition modeling for unstructured parallelized computational fluid dynamics codes,
799 *AIAA journal*, 47, 2894–2906, 2009.

800 Leishman, G. J.: Principles of helicopter aerodynamics with CD extra, Cambridge university press, 2006.

801 Leishman, J.: Modeling sweep effects on dynamic stall, *Journal of the American Helicopter Society*, 34, 18–29, 1989.

802 Letzgus, J., Dürrwächter, L., Schäferlein, U., Keßler, M., and Krämer, E.: Optimization and HPC-Applications of the Flow Solver FLOWER,
803 in: *High Performance Computing in Science and Engineering' 17*, pp. 305–322, Springer, 2018.

804 Levy, D. W., Laffin, K. R., Tinoco, E. N., Vassberg, J. C., Mani, M., Rider, B., Rumsey, C. L., Wahls, R. A., Morrison, J. H., Brodersen, O. P.,
805 et al.: Summary of data from the fifth computational fluid dynamics drag prediction workshop, *Journal of Aircraft*, 2014.

806 Lindenburg, C.: Investigation into rotor blade aerodynamics, ECN Report: ECN-C-03-025, 2003.

807 Loganathan, J. and Gopinath, G.: *Advances in Wind Turbine Aerodynamics*, 2018.

808 Mack, L. M.: Boundary-layer linear stability theory, Tech. rep., CALIFORNIA INST OF TECHNOLOGY PASADENA JET PROPULSION
809 LAB, 1984.

810 Masson, C. and Smaïli, A.: Numerical study of turbulent flow around a wind turbine nacelle, *Wind Energy*, 9, 281–298, 2006.

811 McCormick, D.: Boundary layer separation control with directed synthetic jets, in: 38th Aerospace Sciences Meeting and Exhibit, p. 519,
812 2000.

813 Menter, F. R.: Two-equation eddy-viscosity turbulence models for engineering applications, *AIAA journal*, 32, 1598–1605, 1994.

814 Obert, E.: *Aerodynamic design of transport aircraft*, IOS press, 2009.

815 Post, M. L. and Corke, T. C.: Separation control on high angle of attack airfoil using plasma actuators, *AIAA journal*, 42, 2177–2184, 2004.

816 Przewlocki, J.: Numerische Simulation einer vereinfachten Flügel-Rumpf Konfiguration, Master's thesis, University of Stuttgart, Institute of
817 Aerodynamics and Gas Dynamics, 2017.

818 Purser, P. E. and Spearman, M. L.: Wind-tunnel tests at low speed of swept and yawed wings having various plan forms, Tech. rep., National
819 Aeronautics And Space Administration Hampton VA Langley Research Center, 1951.

820 Rahimi, H., Schepers, G., Shen, W. Z., García, N. R., Schneider, M., Micallef, D., Ferreira, C. S., Jost, E., Klein, L., and Herráez, I.:
821 Evaluation of different methods for determining the angle of attack on wind turbine blades with CFD results under axial inflow conditions,
822 arXiv preprint arXiv:1709.04298, 2017.

823 Rumsey, C. L., Neuhart, D., and Kegerise, M. A.: The NASA juncture flow experiment: Goals, progress, and preliminary testing, in: 54th
824 AIAA Aerospace Sciences Meeting, p. 1557, 2016.

825 Sayed, M., Lutz, T., Krämer, E., Shayegan, S., Ghantasala, A., Wüchner, R., and Bletzinger, K.-U.: High fidelity CFD-CSD aeroelastic
826 analysis of slender bladed horizontal-axis wind turbine, in: Journal of Physics: Conference Series, vol. 753, p. 042009, IOP Publishing,
827 2016.

828 Schepers, J. and Snel, H.: Model experiments in controlled conditions, ECN report, 2007.

829 Schepers, J., Boorsma, K., Cho, T., Gomez-Iradi, S., Schaffarczyk, P., Jeromin, A., Lutz, T., Meister, K., Stoevesandt, B., Schreck, S., et al.:
830 Final report of IEA Task 29, Mexnet (Phase 1): analysis of Mexico wind tunnel measurements, IEA, 2012.

831 Schreck, S. and Robinson, M.: Rotational augmentation of horizontal axis wind turbine blade aerodynamic response, Wind Energy, 5, 133–
832 150, 2002.

833 Schreck, S., Fingersh, L., Siegel, K., Singh, M., and Medina, P.: Rotational augmentation on a 2.3-MW rotor blade with thick flatback airfoil
834 cross sections, in: Proceedings of the 51st AIAA Aerospace Sciences Meeting, AIAA 2013, vol. 915, 2013.

835 Seifert, A., Bachar, T., Koss, D., Shephelovich, M., and Wygnanski, I.: Oscillatory blowing: a tool to delay boundary-layer separation,
836 AIAA journal, 31, 2052–2060, 1993.

837 Simpson, R. L.: Junction flows, Annual Review of Fluid Mechanics, 33, 415–443, 2001.

838 Snel, H., Houwink, R., Bosschers, J., Piers, W., Van Bussel, G., and Bruining, A.: Sectional prediction of 3D effects for stalled flow on rotating
839 blades and comparison with measurements, in: Proc. European Community Wind Energy Conference, HS Stevens and Associates, LÅ1,
840 vol. 4, 1993.

841 Sørensen, N., Hansen, M., Garca, N., Florentie, L., Boorsma, K., Gomez-Iradi, S., Prospathopoulos, J., Papadakis, G., Voutsinas, S., Barakos,
842 G., et al.: Power curve predictions wp2 deliverable 2.3, Technical Report, 2014.

843 Sørensen, N., Garca, N., Voutsinas, S., Jost, E., and Lutz, T.: Aerodynamics of Large Rotors WP2 Deliverable 2.6 Effects of complex inflow
844 for the AVATAR reference rotor and NM80 rotors, Technical Report, 2017.

845 Spalart, P. and Allmaras, S.: A one-equation turbulence model for aerodynamic flows, in: 30th aerospace sciences meeting and exhibit, p.
846 439, 1992.

847 Spalart, P. R.: Strategies for turbulence modelling and simulations, International Journal of Heat and Fluid Flow, 21, 252–263, 2000.

848 Tanner, M.: Reduction of base drag, Progress in Aerospace Sciences, 16, 369–384, 1975.

849 Thwaites, B.: Approximate calculation of the laminar boundary layer, The Aeronautical Quarterly, 1, 245–280, 1949.

850 Van Dam, C., Kahn, D. L., and Berg, D. E.: Trailing edge modifications for flatback airfoils, SAND2008-1781, Sandia National Laboratories,
851 Albuquerque, NM, 2008.

852 Vassberg, J., Tinoco, E., Mani, M., Rider, B., Zickuhr, T., Levy, D., Brodersen, O., Eisfeld, B., Crippa, S., Wahls, R., et al.: Summary of the
853 Fourth AIAA CFD Drag Prediction Workshop (2010). AIAA Paper No, Tech. rep., AIAA-2010-4547, 2010.

854 Vassberg, J. C., Tinoco, E. N., Mani, M., Brodersen, O. P., Eisfeld, B., Wahls, R. A., Morrison, J. H., Zickuhr, T., Lafin, K. R., and Mavriplis,
855 D. J.: Abridged summary of the third AIAA computational fluid dynamics drag prediction workshop, Journal of Aircraft, 45, 781–798,
856 2008.

- 857 Weihing, P., Letzgus, J., Bangga, G., Lutz, T., and Krämer, E.: Hybrid RANS/LES capabilities of the flow solver FLOWer-application to flow
858 around wind turbines, in: *The 6th Symposium on Hybrid RANS-LES Methods*, Strassbourg, 2016.
- 859 Weihing, P., Schulz, C., Lutz, T., and Krämer, E.: Comparison of the Actuator Line Model with Fully Resolved Simulations in Complex
860 Environmental Conditions, in: *Journal of Physics: Conference Series*, vol. 854, p. 012049, IOP Publishing, 2017.
- 861 Zahle, F. and Sørensen, N. N.: Characterization of the unsteady flow in the nacelle region of a modern wind turbine, *Wind Energy*, 14,
862 271–283, 2011.
- 863 Zamir, M.: Similarity and stability of the laminar boundary layer in a streamwise corner, *Proc. R. Soc. Lond. A*, 377, 269–288, 1981.
- 864 Zess, G. and Thole, K.: Computational design and experimental evaluation of using a leading edge fillet on a gas turbine vane, in: *ASME*
865 *Turbo Expo 2001: Power for Land, Sea, and Air*, pp. V003T01A083–V003T01A083, American Society of Mechanical Engineers, 2001.

Table C1. Nomenclature

Symbol	Unit	Description
c	$[m]$	Airfoil chord length
C_d	$[-]$	Drag coefficient
$C_{f,x}$	$[-]$	Chord-wise skin friction coefficient
C_l	$[-]$	Lift coefficient
C_p	$[-]$	Pressure coefficient
d_w	$[m]$	Wall distance
L	$[m]$	Length of the nacelle
N	$[-]$	Critical amplification factor of Tollmien-Schlichting waves
p	$[Pa]$	Local static pressure
p_∞	$[Pa]$	Ambient pressure
$q_{\infty,rot}$	$[Pa]$	Kinematic stagnation pressure in the rotating frame
r	$[m]$	Local radius
R	$[m]$	Radius of the blade
u	$[m/s]$	Axial velocity
u_c	$[m/s]$	Velocity in chord-wise direction
u_{mag}	$[m/s]$	Velocity magnitude
U_∞	$[m/s]$	Wind speed
$U_{\infty,rot}$	$[m/s]$	Kinematic inflow velocity in the rotating frame
v	$[m/s]$	Lateral velocity in the rotating frame
w	$[m/s]$	Vertical velocity in the rotating frame
$\langle vw \rangle$	$[m^2/s^2]$	Reynolds shear stress
\tilde{w}	$[m/s]$	Vertical motion relative to the ideal circular path.
x	$[m]$	Axial coordinate
y	$[m]$	Lateral coordinate in the rotating frame
y_p^+	$[-]$	Non-dimensional wall distance wall adjacent cell
z	$[m]$	Vertical coordinate in the rotating frame
x_c	$[m]$	Coordinate in chord-wise direction
α	$[-]$	Angle of attack
Γ	$[m^2/s]$	Bound circulation
η	$[m]$	Blade-normal coordinate in the corner frame
ρ	$[kg/m^3]$	Density
Ω	$[1/s]$	Rotational speed
ω	$[1/s]$	Vorticity
ζ	$[m]$	Vertical coordinate in the corner frame