Integration of System Level CFD Simulations into the Development Process of Wind Turbine Prototypes

Matthias Arnold¹, Florian Wenz², Timo Kühn¹, Thorsten Lutz², Andree Altmikus¹

¹ Wobben Research and Development GmbH, Borsigstr. 26, 26607 Aurich, Germany
² Institute of Aerodynamics and Gas Dynamics (IAG), University of Stuttgart, 70569 Stuttgart, Germany

E-Mail: Matthias.Arnold@Enercon.de

Abstract. During the development process of wind turbines, there is a rising demand for detailed flow and multi-physics analyses. Especially the system level computational fluid dynamics (CFD), focusing not on a single component or aspect, but on the turbine as a system of geometric and dynamic properties can be used to improve the turbine quality and time to market. In this paper the integration of the system level CFD into the development process of Enercon wind turbines is introduced. Further two example applications are detailed presenting the benefits of CFD based nacelle anemometer calibration and a risk assessment study based on aero-servo-elastic CFD simulations.

1. Introduction

Based on the need for low cost of energy on the wind energy market, the development of a wind turbine requires highly reliable simulation and design tools to prevent cost and time intensive iterative procedures. Following this premise of first-time-right, a fast, efficient and flexible simulation environment is required, capable of reducing risks and showing the potential of interaction between different physical topics. Subsequently, many simulation approaches developed in academic research are integrated into the industrial application. Among those, especially the aerodynamics as source of power production and disturbances or loads are an important aspect of such a simulation environment.

The present publication will introduce the transfer of full turbine and aero-elastic computational fluid dynamics (CFD) applications from academic simulations into the industrial workflow of turbine and prototype development. The benefits of such simulation results and effort required for a system level CFD simulation of a wind turbine, including rotor blades, nacelle, tower, add-ons, atmospheric conditions, aero-servo-elastic responses, etc. will be discussed. Within this discussion first the implementation into the product development process will be shown, followed by the detailed presentation of two exemplary applications.

2. Objectives and Methodology

In contrast to the academic point of view, where the focus often lies on selected highly resolved cases, e.g. [2], [10], with much effort invested into an optimal setup and detailed phenomenological studies, the industrial focus is on minimal effort for the evaluation of a large number of different cases with an
allowance for non-optimal computational efficiency. Thus a robust setup and automatization is required.

2.1. System Level CFD Simulation

Based on this requirement from the industrial processes, a framework for system level CFD has been defined. System level CFD is defined here to be a computational fluid dynamics simulation incorporating the whole system of a wind turbine into the simulation. The system level CFD is therefore an extension of the standard steady rigid rotor blade simulation with further geometric components, environmental conditions, structural properties, cf. [10], surface treatments, operational points, etc. as sketched in Figure 1. This approach does not mean that all those aspects are always part of the simulation, but that each additional detail is a module, which can be selected to be activated based on the requirements. In Figure 1 this module selection is sketched for the application examples presented later in section 5 and section 6.

![Figure 1. Overview of the modularized system level CFD framework (no claim to completeness).](image)

2.2. CFD-Solver

In the present example of system level CFD simulations the task is addressed by URANS CFD simulations with the solver FLOWer developed by the German Aerospace Center DLR, [16]. FLOWer is a block-structured, compressible code with the chimera grid overset method. Enercon adapted this code to industrial needs by including additional in-house modules and modules developed by the Institute of Aerodynamics and Gas Dynamics at the University of Stuttgart, [9], for wind energy applications. This setup allows for flexible adaption of the model while offering high quality results.

2.3. Automatization of Methods

The main cost driver in the setup of block-structured CFD simulations is meshing. By developing a robust automated meshing algorithm based on Pointwise-TCL-scripting the meshing effort is limited. Robust means in this context, that the algorithm automatically adapts to e.g. blunt, sharp and cylindrical trailing edges or different blade tip shapes.

Within development of such an algorithm it is essential to avoid decision points (if-clause) based on grid or geometry properties. As those properties change over e.g. the rotor radius, jumps in the grid distribution with subsequent locally insufficient quality might occur. Therefore, blending between methods within the algorithm is required. The algorithm itself can be for example a blade grid based on single radial layers. This approach allows for a local adaption to e.g. trailing edge angles. However, the layers need to be coupled by filtering functions based on the radial distance of the layers to again avoid locally insufficient quality.

Such an algorithm was developed covering a large variety of blade shapes as presented in [4]. Nevertheless, for a robust procedure it is essential to integrate automatic quality checks and quality based errors otherwise weak spots in the grid quality might pass silent on an automatic process spoiling the result.
Besides the automatic meshing, the second big step to limit the manual intervention is the permutation and processing of cases. For this step a Python-based environment has been developed which allows for automatic setup, permutation, processing and post-processing of cases also presented in [4]. Also within this environment the approach of modularization has been applied. Thus, modules for all usual data analysis on wind turbine related CFD simulations are available as distribution of induction, local loading, separation regions, angles of attack (cf. [9]), 3D airfoil polars, etc. and can be easily extended by case specific modules.

With the modularization and those automatization blocks in place the system level CFD simulations are a streamlined process, which enables the development team to analyze new geometry variations, structural properties, operational points or atmospheric conditions with minimal effort and guarantees high quality.

2.4. Generic I-82 wind turbine

The investigations shown in this publication are based on a generic turbine model. This I-82 turbine has been developed in the research Project IndiAnaWind [3], as a generic derivate of the Enercon E-82 turbine. The setup of the I-82 has been tuned to provide comparable aerodynamic, elastic and servo-dynamic properties as the E-82. However, as a generic turbine it is neither identical, nor features the detailed properties. The basic parameters of the I-82 are given in Table 1. The steady points of operation of the turbine are shown in Figure 4.

For the controller of the turbine a mixture of literature definitions from [5], [6], [8], [15] have been used, which reproduces the operational properties of the E-82. However, none of the industrial dynamic load mitigation methods of the E-82 have been transferred.

Table 1. Parameters of generic I-82 turbine

| Parameter                | Value               |
|-------------------------|---------------------|
| Rotor diameter          | 82m                 |
| Hub height              | 97m                 |
| Rated power             | 2.365MW             |
| Rated rotational speed  | 18rpm               |
| Rated wind speed        | 14m/s               |
| Controller              | Variable speed and pitch with peak shaving |
| Blade shape             | airfoil shape starting at hub and winglet at tip |
| Nacelle shape           | Ellipsoid           |

For the present applications the turbine has been simulated with FLOWer in the URANS mode with the SST-turbulence modeling, [14]. Figure 2 shows the nacelle and blade grid structure with the overset grid regions and the hanging grid node structures in the wake.

Figure 2. Block structured surface and blade grid for the generic I-82 turbine.

Grid quality, numerical settings and convergence follow the state of the art requirements for CFD simulations. The grid resolves the boundary layers with a $y^+ \approx 1$, approx. $11 \cdot 10^6$ Cells on each rotor
blade grid. The farfield boundary condition is with \( \geq 10 \) rotor radius far enough away from the turbine to prevent disturbances on the results. Case specific details are given below.

3. Application within the development process

The modularized and automated concept for system level CFD is part of the turbine and prototype development process for each Enercon Wind turbine. Within the process, the CFD is bound to three phases in the process as shown in Figure 3: Design, Installation and Testing.

![Figure 3. Prototype timeline with points of CFD integration.](image)

This scheme applies for the whole span of development processes from new turbine generations down to smaller technical changes. However, single steps of the CFD augmentation in the process might be skipped if not required.

In the following sections an exemplary excerpt of applications of the system level CFD is presented. Each of those applications is an established part of the Enercon product development process, used on a daily basis and results in an increased understanding of the system and thus improvement of the system during development.

4. Load Estimation for add-ons and attached equipment

One of the basic applications of system CFD-simulations is the evaluation of loads on specific components. By introducing those geometrically or virtually into the simulation the impact of local flow phenomena, e.g. hub vortex, partial shadowing, etc., see Figure 5, can be evaluated. The resulting loads can then be used to calculate the required structural integrity, but also define limiting values for the stiffness regarding aeroelastic stability of components. This type of simulation has a broad range of levels of detail, but is typically a subset of the applications shown below.

5. Anemometer Calibration

The alignment of a turbine to the incoming wind direction is controlled for most modern wind turbines based on the nacelle anemometer. The nacelle anemometer is the only device being able to give robust measurements of the incoming wind under all operational and non-operational conditions. However, as the anemometer operates in the near wake of the rotor, the calibration of this sensor is essential.

On commercial Enercon turbines this calibration is done in two steps: first a pre-calibration curve based on reference turbine measurements is applied and in the second step this curve is fine-tuned and site calibrated based on specific machine learning algorithms and filters, incorporating further operational data of the turbine for an optimal alignment of the turbine at each site. However, for the prototype itself the input for the first step of this process is not available initially as the prototype is the first of its kind.

5.1. Data driven anemometer calibration procedure

The classical approach for calibrating the anemometer on the prototype was in the past based on iterative measurements. Starting with reference data of another turbine, the turbine was commissioned
and the nacelle orientation measured compared to a met mast. This offset was recorded for the full operational range, which takes some time. The offset curve was then added to the initial calibration curve. However, the power of the turbine, point of operation and aerodynamics also differ under crossflow operation. Thus, the offset curve is not the full error in the calibration curve and the process needs to be repeated multiple times to remove the influence of yaw-misalignment to the calibration step by step. This iterative process took several weeks before the power and load measurement could be started.

5.2. Simulation driven anemometer calibration procedure

By applying the system level CFD in the modern CFD-integrated workflow, the pre-calibration curve for the anemometer is simulated before installation of the first prototype. This is done by unsteady RANS CFD-simulations of the turbines surface including the three blades, tower, nacelle, etc. on quasi-steady points of operation. A 120°-CFD-Setup which could be simulated with steady RANS is not suitable here, as the impact of tilt and tower on the anemometer are too large. Structural deformations are irrelevant, as the impact of the blade root aerodynamics is dominant for the anemometer inflow and not suspect to large displacements.

Thus, by extending the numerical model used previously in the development process for load estimation into the required resolution, the pre-calibration curve of the anemometer can be calculated and the iterative measurement process can usually be avoided. The site-specific fine-tuning is then done as described above with the machine learning algorithm.

5.3. Simulation results

A total of 13 points of operation have been simulated for the application case of anemometer calibration for the generic I-82 turbine as shown in Figure 4. Each point of operation was simulated with constant homogeneous inflow for 13 revolutions till the wakes were fully evolved and a repetitive pattern in all flow parameters was confirmed. This pattern however is not completely periodic as vortex shedding occurs with a frequency diverging from the blade passing frequency. This vortex shedding is visible in Figure 5 on the tower and the blade root for an exemplary point of operation.

Based on the grid size of $55.4 \cdot 10^6$ cells and the time step resolution of $2^\circ$, each simulation requires approx. $16.3 \cdot 10^3$ CPUh to reach converged statistics of the solution.

By extracting the velocity components at the anemometer position $4m$ above the intersection of tower and rotor axis, the local time dependent flow direction can be calculated. As shown in Figure 6 this flow direction oscillates with a repetitive pattern with a dip in the angle each time a blade passes the anemometer. However, the before mentioned vortex shedding at the blade root interferes with the anemometer causing additional fluctuation on the local flow direction. The significance of this disturbance rises with strong separation at the blade root.
Nevertheless, by evaluating a long enough time period and applying an appropriate averaging method, a mean offset of the direction from 0° can be identified. This offset results from the local power production of the rotor blade, causing the wake into a counter-rotating swirl. Without a correction of the measured anemometer direction, the nacelle would be misaligned to the wind by the yaw-controller. A similar correction also applies for the velocity which is disturbed by the induction of the rotor as shown by e.g. [7] and [13].

Therefore, based on the CFD results correction values for the anemometer data are introduced as $v_{\text{corrected}} = k_{\text{anemo}} \cdot v_{\text{measured}}$ for the velocity and $\varphi_{\text{corrected}} = \varphi_{\text{measured}} - \beta_{\text{anemo}}$ for the direction depending on the point of operation, Figure 7. The resulting curves are not completely smooth based on the above described dependency on the vortex size. Especially at the transition between peak shaving and speed limitation around 12 m/s the position of the hub vortex moves inwards, causing a local change in the correction factors. Nevertheless, with those corrections in place, the turbine can be aligned to the undisturbed wind direction based on the nacelle anemometer data.

Figure 7. Velocity and direction corrections for the generic I-82 turbine.

5.4. CFD augmented prototype commissioning
The discussed method of pre-calibration of the nacelle anemometer with simulations has been incorporated into the commercial prototype commissioning process of several state of the art turbines with different nacelle shapes and anemometer setups. During those applications the CFD-based precalibration method proved to provide an accuracy with an error < 2° compared to field measurements, which is well within the range of the machine learning algorithm for fine tuning of the calibration. An iteration of the calibration curve was therefore avoided based on the CFD results and
the measurement campaigns were accelerated by approx. 4-6 weeks compared to a commissioning without CFD augmentation. This time advantage translates directly into an equivalently improved time to market.

6. Numerical Forensics

During the design of a turbine a large variety of wind conditions and events are simulated with design methods like the Blade-Element-Momentum-Theory. However, the analysis of three dimensional, highly transient or multi-physic effects is limited with such methods. Thus, based on the limitations of those tools the investigations are extended or solely performed with the system level CFD.

Such investigations appear in two forms: Preemptive investigations during development typically raising a question of the type: “What happens to the system if ...?” and root cause analysis type investigations asking for: “What happened when ...?”. Both forms require the same steps of investigation and are thus summarized into the simulation category of numerical forensics.

6.1. Steps on a numerical forensics investigation

Any numerical forensics analysis starts with the problem identification. This step should result in a thorough description of the case including relevant turbine and subsystem properties, operational parameters, meteorological conditions, site reports if available, etc. Besides those descriptive data, within the problem identification also reference field or simulation data of other turbines is essential. This data is used to identify the differences between the condition to analyze and conditions known to cause effects or being unproblematic.

Based on the identified problem an estimation of the system behavior is required. This step requires expert knowledge and experience and is a typical step in a risk assessment. In contrast to a normal risk assessment process, this estimation is not taken as final result, but revised based on the detailed investigation. However, an inaccuracy in this step might lead to increased resource requirements in the further process.

From the expert estimation the required level of detail for the investigation is deduced and simulations of the conditions at question are performed. Those steps are repeated in an iterative manner based on the rising knowledge.

After identifying the system behavior, approaches for mitigating the risks if required can be applied. Also the results must be introduced to the knowledge database for any further numerical forensics analysis or system design.

6.2. Application case - Timing of shutdown after a grid loss event

As an exemplary application case of such a numerical forensics investigation, different shutdown scenarios for the generic I-82 turbine introduced above are investigated. This investigation has no field equivalent, but demonstrates the approach to the numerical forensics analysis well. The hypothetic question of concern for the example application case is: “Is there any aero-elastic threat to the blade for the generic I-82 turbine, if a grid loss would remain undetected by the system? Is there a critical time for the detection?”

Such a preemptive question and analysis is typically part of the risk assessment during the design of a turbine or in the preparation of a field test.

6.2.1. Required level of detail: Following the steps given in section 6.1 first the required level of detail is defined. For the analysis the generic I-82 turbine presented in section 2.4 is used. The main influencing parameters for the investigated event are the dynamic rotor blade aerodynamics and the blades structural flexibility. Due to the short time scales of the shutdown, turbulent and inhomogeneous inflow conditions are not expected to have a significant impact and are therefore neglected. The same accounts for the tower flexibility, which will influence the overall loads, but due to the very low eigenfrequency compared to the blade dynamics is not expected to have a significant
6.2.2. Case variation: To distinguish between the effects of a grid loss shutdown and a delayed detection of the grid loss, four different types of simulations are run. Those are a normal shutdown and a grid loss shutdown with immediate detection to identify the differences due to the generator being part of the braking procedure, as well as grid loss shutdowns with a medium delay of 0.5 s and a delay long enough to show whether the turbine would settle at a new point of operation or diverge. The latter delay case represents an undetected grid loss as requested by the scenario.

6.2.3. Results of the study: The basic behaviour of the turbine during the four different shutdown scenarios is shown in Figure 8. All simulations start at the settled point of operation for a wind speed of 10 m/s with approx. 16.33 rpm and 2.5° pitch. For the normal shutdown, the pitch angle starts at $t = 0$ s to increase with a rate of +5°/s. However, the aerodynamic response delays the change in aerodynamic power and thus the reduction of operational speed by approx. 1.5 s. After that time, for a normal shutdown including the generator torque, the rotational speed is smoothly reducing.

In contrast to that, the brake down of the generator torque upon the simulated grid loss causes the turbine to speed up. The aerodynamic power remains almost unchanged for the first second after the event. This allows the rotor to speed up to approx. 17.6 rpm before the change in pitch angle reduces the power and subsequently the rotational speed. The negative acceleration of the generator speed is in the same order of magnitude as on a normal shutdown including the generator torque. This shows, that the dominant decelerating effect for the rotor is the blade aerodynamics.

However, the rotational speed shows ripples in the process, which were not apparent for the normal shutdown. Those result from the rapid unloading of the generator, exciting the edgewise respectively inplane and higher order modes of the blade structure. This oscillation is damped over time.

Considering a delay in the detection of the grid loss, the behavior of the system changes. As seen on the detected grid loss, the rotational speed of the rotor increases but remains now for a longer time on a higher rotational speed, before the pitch starts to increase towards 90°.

It is significant here, that even without detecting the grid loss initially, the pitch starts to increase shortly after the event. This is caused by the normal rotational speed control loop of the generic turbine controller, which limits the rotational speed to the rated speed of the turbine even without the generator. Thus, also with a very long delay in the detection of a grid loss, the turbine does not go into an overspeed condition, but settles for a point of operation around rated rotational speed. However, the damping of the rotational speed by the controller is weak compared to normal operation, as the speed controller is designed incorporating the damping effect of the generator.

In Figure 9 the blade loads between the four shutdown scenarios are compared for the representative flapwise direction. As can be seen for the flapwise blade tip deflection and blade root bending moment, in all four scenarios the blade loads are significantly reduced after the event at $t =$
The turbine controller is pulling the turbine into a save state even without detecting the cause for the change in behavior. There are no unusual loads apparent in the simulation.

Figure 8. Rotational speed (left) and pitch angle (right) over time on different shutdown scenarios for the generic I-82 turbine.

Figure 9. Blade tip flapwise deflection (left) and flapwise blade root bending moment (right) over time on different shutdown scenarios for the generic I-82 turbine.

Based on this analysis the initial question of the presented application case for numerical forensics can be answered as: There is no aero-elastic threat to the rotor blade of the generic I-82 turbine in a grid respectively generator loss scenario. Even if the event passes undetected, the turbine settles on a stable point of operation well within normal operational parameters, given that the controller remains operational.

7. Outlook to future development of the system level CFD simulation

Despite the large steps done in the past and the integration of the system level CFD into the design process, there is a need for further development of the modularization of multi-physic-simulation-methods and the process automatization. However, the possibilities of the modularized simulation methods for wind energy applications extend far beyond the current state of the art and there are still open questions regarding the combined effect of phenomena, which need to be addressed in future research. E.g. what is the interaction between dynamic stall and the controller for the simulation of aeroacoustics? What is the impact of aeroelasticity and freestream turbulence on the anemometer calibration? Is the RANS/URANS feasible for future development? How can small scale geometric structures like vortex generators and serrations be incorporated into large simulation frameworks? Such questions will be addressed in the research project IndiAnaWind, recently started in autumn 2019.
8. Conclusions
In this publication the industrial application for wind turbine system level CFD is discussed and the implementation into the turbine design process is shown. The method and its challenges as well as the worth and effort of system level CFD are analyzed from an industrial point of view. The paper presents application examples of load simulations, simulation based nacelle anemometer calibration and aero-servo-elastic coupled CFD with variable speed and pitch for numerical forensics. Combined, those applications of system level CFD can be used to increase the quality of the product during development and shorten the time to market for new turbines by several weeks.

Acknowledgments
This research was partially funded by the German Federal Ministry for Economic Affairs and Energy in the research project IndiAnaWind (“Interdisziplinäre Analyse und Optimierung von Windenergieanlagen und ihrer Komponenten”) under grant number 0325719E.

References
[1] Hansen A, Cutululis N, Sørensen P, Iov F, Larsen T, “Simulation of a flexible wind turbine response to a grid fault” at EWEC 2007, Milan, Italy
[2] Arnold M, “Simulation and Evaluation of the Hydroelastic Responses of a Tidal Current Turbine”, PhD-Thesis, 2016, ISBN 978-3-8439-3199-1
[3] Arnold M, “Definition der generischen Windenergieanlage ‘I82’ für das Projekt IndiAnaWind mit Aero-Servo-Elastischer-Ähnlichkeit zur WEA Enercon E82“, Del. 1.2, Research Project IndiAnaWind, 2019
[4] Arnold M, et al., “Schlussbericht Smartblades 2 – WRD – öffentlicher Teil“, Research Project Smartblades 2, FKZ 0324032, to be published in 2020
[5] Bak C, et al., “The DTU 10-MW Reference Wind Turbine“, DTU, 2013
[6] Fischer B., Shan M, “A Survey on control methods for the mitigation of tower loads”, IWES Project report 01/104256, 2013
[7] Frandsen S, Sørensen J, Mikkelsen R, Pedersen T, “The generics of wind turbine nacelle anemometry” at EWEC 2009, Marseille, France
[8] Jonkman J, Butterfield S, Musial W, Scott G, “Definition of a 5-MW Reference Wind Turbine for Offshore System Development", NREL/TP-500-38060, 2009
[9] Jost E, Klein L, Leipprand H, Lutz T, Krämer E, “Extracting the angle of attack on rotor blades from CFD simulations”, in Wind Energy, Vol. 21 (2018), pp. 807-822, doi: 10.1002/we.2196
[10] Klein L, Gude J, Wenz F, Lutz T, Krämer E, “Advanced computational fluid dynamics (CFD)-multi-body simulation (MBS) coupling to assess low-frequency emissions from wind turbines” in Wind Energy Science, Vol.3 (2018) pp. 713-728, doi: 10.5194/wes-3-713-2018
[11] Kühn T, et al., “Results of the research project AssiSi” in Journal of Physics: Conference Series 1037 (2018) 022040, doi: 10.1088/1742-6596/1037/2/022040
[12] Li Y, Castro A, Martin J, Sinokrot T, “Coupled computational fluid dynamics/multibody dynamics method for wind turbine aero-servo-elastic simulation including drivetrain dynamics” in Renewable Energy, Vol. 101 (2017), pp. 1037-1051, doi: 10.1016/j.renene.2016.09.070
[13] Martin C, Lundquist J, Clifton A, Poulos G, Schreck S, ”Atmospheric turbulence affects wind turbine nacelle transfer functions” in Wind Energ. Sci., Vol. 2 (2017), pp. 295-306, doi:10.5194/wes-2-295-2017
[14] Menter F: “Two-Equation Eddy-Viscosity-Transport Turbulence Models for Three-Dimensional, Shock-Separated Flows” in AIAA Journal, Vol. 34 (1994), pp. 756-763
[15] Popko W, et al., “IWES Wind Turbine IWT-7.5-164 – Rev. 4”, IWES, 2018
[16] Raddatz J, Fassbender JK, “Block-Structured Navier-Stokes Solver FLOWer” in Notes on Numerical Fluid Mechanics and Multidisciplinary Design, Vol. 89 (2002), pp. 27-44, doi: 10.1007/3-540-32382-1_2