Journal of Physics: Conference Series

Results of the research project AssiSt

To cite this article: Timo Kühn *et al* 2018 *J. Phys.: Conf. Ser.* **1037** 022040

The Science of Making Torque from Wind (TORQUE 2018) IOP Publishing
IOP Conf. Series: Journal of Physics: Conf. Series **1037** (2018) 022040 doi:10.1088/1742-6596/1037/2/022040

Results of the research project AssiSt

Timo Kühn[1], Andree Altmikus[2], Hussam Daboul[2], Alexander Radi[2], Siegfried Raasch[3], Christoph Knigge[3], Lennart Böske[3], Thorsten Schwarz[4], Christoph Heister[4], Annika Länger-Möller[4], Thorsten Lutz[5], Pascal Weihing[5], Christoph Schulz[5], Jakob Thiemeier[5], Charles Mockett[6], Marian Fuchs[6] and Frank Thiele[6]

[1] Wobben Research & Development GmbH, Teerhof 59, 28199 Bremen, Germany
[2] Wobben Research & Development GmbH, Borsigstr. 26, 26607 Aurich, Germany
[3] Institute of Meteorology & Climatology, Leibniz University Hannover, Herrenhäuser Str. 2, 30419 Hannover, Germany
[4] Institute of Aerodynamics and Flow Technology, German Aerospace Center (DLR), Lilienthalplatz 7, 38108 Braunschweig, Germany
[5] Institute of Aerodynamics and Gas Dynamics, University of Stuttgart, Pfaffenwaldring 21, 70569 Stuttgart, Germany
[6] CFD Software E+F GmbH, Charlottenburger Innovations Centrum (CHIC), Bismarckstraße 10-12, 10625 Berlin, Germany

E-Mail: [1,2]{timo.kuehn, andree.altmikus, hussam.daboul, alexander.radi}@enercon.de, [3]{raasch, knigge}@muk.uni-hannover.de, [3]thorsten.schwarz@dlr.de, [4]{lutz, weihing, thiemeier}@iag.uni-stuttgart.de, [5]{charles.mockett, marian.fuchs, frank.thiele}@cfd-berlin.com

Abstract. This article gives an overview of the results of the wind energy research project AssiSt. Results of the four work packages include flow in complex terrain, wind energy converters (WEC) in complex terrain subject to atmospheric inflow, laminar-turbulent transition, generator cooling, hub aerodynamics, and passive flow control devices. Four different flow solvers (PALM, FLOWer, THETA, OpenFOAM) are in use during the course of the project depending on the corresponding problem requiring specific solver features. Key achievements of the project are the coupling of atmospheric LES (PALM) and URANS simulations of the complete WEC (FLOWer as well as THETA) in order to impose the external turbulent flow fields to the inflow of the WEC for physics resolved load simulations, numerical replication of complex flows over blades with vortex generators using OpenFOAM and efficiency-augmented pressure loss simulations on very complex industrial geometries of ENERCON's direct drive WEC generators for precise cooling analyses. Industrial validity of all methods, model and process developments were key objectives of this research project.

1. Introduction

Wind energy converters (WECs) are complex technical engines characterized by strong multi-physical interaction within their surrounding environment. Simulation tools for the analysis of technical details such as structure, aerodynamics, meteorology or electrical engineering, to name only the most evident disciplines, play an essential role in the design of WECs due to their high flexibility and usability at early design stages. Because of the wind turbine's distinct purpose to transform wind into energy, aerodynamics and thus Computational Fluid Dynamics (CFD) inherit a prominent role. Enercon, represented in the project by its Research & Development affiliation *Wobben Research and Development* (WRD), as one of the leading manufacturers of wind turbines collaborated in the publicly funded research project AssiSt with the *Institute of Aerodynamics and Gas Dynamics* (IAG) of the University of Stuttgart, the *Institute of Meteorology and Climatology* (IMUK) of the Leibniz University in Hannover, the *German Aerospace Center* (DLR) embodied by the Institute of Aerodynamics and Flow Technology in Braunschweig and the engineering consultancy *CFD Software Entwicklungs- und Forschungsgesellschaft GmbH* in Berlin (CFDB) in order to develop and improve several CFD codes for applications concerning wind energy. The project's major objective was to incorporate all features of aerodynamic relevance, which includes the simulation of the complete exterior of the turbine with and without atmospheric inflow and in complex terrain, the interior of the generator, and finally the meteorological environment.

These topics were addressed in 4 work packages: WP1 (Wind in Complex Terrain), WP2 (Complete Wind Turbine), WP3 (Future Methods in Rotor Simulation), and WP4 (Efficiency Improvement).

2. Simulation Software

As all project members are representatives of the Computational Fluid Dynamics community and as the clear intention of the project was to strengthen the application of numerical tools in the design and revision of wind energy converters, an intrinsic motivation of the project partners was to enhance and demonstrate the potential of their most established CFD tool set. Each solver has its particular development history and is thus preferably used in certain fields; not implying that it is limited to this. The four CFD codes used during the course of AssiSt will be shortly described here as follows. The employed physical modelling fidelity ranges from standard (Unsteady) Reynolds Averaged Navier Stokes ((U)RANS) methods via Detached Eddy Simulation (DES) to Large Eddy Simulation (LES).

2.1. PALM

More than 15 years of development and practical use at IMUK makes the PArellelized LES Model (PALM) [1] one of the world's leading meteorological turbulence resolving simulation models. Besides extensive environmental applications by numerous research groups worldwide (e.g. [2]), the code was already in use for wind-energy relevant applications [3]. More recently, substantial improvements were implemented to address more practical issues, for example the implementation of topography, non-cyclic inflow boundary conditions featuring turbulence recycling and the incorporation of a vegetation model. Since 2012 PALM is available under the GNU Public License (GPL) v3. The model PALM is based on the non-hydrostatic, filtered, incompressible Navier-Stokes equations in Boussinesq-approximated form. By default, PALM has at least six prognostic quantities: the velocity components, the potential temperature, water vapor mixing ratio and optionally a passive scalar. Furthermore, an additional equation is solved in the LES mode for the subgrid-scale turbulent kinetic energy (SGS-TKE). By default, the advection terms in the prognostic equations are discretized using an upwind-biased 5th-order differencing scheme in combination with a 3rd-order Runge–Kutta time-stepping scheme. A predictor corrector method is used where an equation is solved for the modified perturbation pressure after every time step to encounter the presence of flow divergence due to the incompressibility limitation of the Boussinesq approximation.

2.2. FLOWer

The FLOWer code, initially developed by a joint team from DLR, universities and partners within the aerospace industry, has its roots in fixed wing and helicopter aerodynamics. Due to its comprehensive overset meshing capabilities the code proved to be an appropriate software environment for rotating systems such as wind energy converters. For the last couple of years IAG provides the most active developer group of FLOWer with focus on helicopter and wind energy aerodynamics [4]. A well-proven process chain, originating from several research projects, allows for the simulation of a wind energy power plant in turbulent atmospheric inflow. Validation of this process chain was done within the MexNext Project [5] via comparison with PIV and load measurements. FLOWer is a finite volume, block-structured RANS solver. The convective fluxes are approximated either via a central discretization scheme with artificial viscosity or upwind schemes. Time integration is performed using explicit multistage time-stepping schemes. For steady calculations convergence is accelerated by implicit residual smoothing, local time stepping and multigrid. For time accurate calculations an implicit time integration according to the dual time stepping approach is employed. FLOWer handles standard multiblock meshes, meshes with hanging nodes at block interfaces and overlapping blocks based on the Chimera technique. Several transition prediction methods are integrated as well as a number of transport equation turbulence models and DES capabilities. A significant extension was implemented by IAG in terms of a fifth order WENO scheme to improve preservation of vortical structures.

2.3. THETA

THETA and its close relative TAU are the present DLR in-house CFD flow solvers of the post-FLOWer era. In contrast to FLOWer, both codes pursue an unstructured, cell-vertex mesh strategy. Both codes share a large number of peripheral code structures, but TAU’s kernel solves the RANS equations in compressible form while THETA, originally conceived as a heat transport and combustion extension to TAU [6], is purely incompressible. THETA was significantly extended at DLR for usage in the field of wind energy science in the national research project SimBA. THETA uses a 3D finite volume method on unstructured hybrid grids with a dual grid approach. The Poisson equation for the pressure velocity coupling is solved either via a SIMPLE algorithm or a projection method. Time discretization can be done with a couple of implicit, semi-implicit and explicit methods. A large number of turbulence models and DES/LES options are implemented. THETA profits from the elaborate development of TAU in terms of the large number of available code extensions and add-ons such as overset mesh capabilities, fluid-structure interfaces, mesh input and output filters, mesh adaptation, and optimization features.

2.4. OpenFOAM®

The community solver OpenFOAM [7] is currently possibly the most widely used open source CFD solver package in academia and industry. It is licensed under the GNU Public License v3 standard. In particular the wide variety of extensions (e.g. particle tracer, multi-phase flow, electromagnetics) and the source code availability add to the increasing dissemination in the private, academic and industrial environment. OpenFOAM offers built-in mesh generation tools, one of which (snappyHexMesh) is applied in WP4 of the AssiSt project. In this project context, the incompressible version of the cell-centered finite-volume code with pressure velocity coupling was preferred. For spatial discretization, either 2nd order accurate upwind or central differencing schemes are used, dependent on the chosen turbulence modelling approach. RANS and DES-based turbulence modelling approaches have been applied, the latter using thoroughly-validated implementations that were contributed to the central OpenFOAM version by CFDB. For all unsteady simulations, an implicit 2nd order accurate backward Euler scheme was applied.

3. Results

Each work package had a preparatory phase, where industrial requirements and prerequisites were defined and a validation/verification phase in which the final numerical results were either compared to measurement data or data generated from established industrial tool chains.

3.1. Results of Work Package 1

Work package 1 was subdivided into 4 Tasks: WP1.1 (Complex Terrain), WP1.2 (Ground Boundary Conditions), WP1.3 (Velocity Fields), and WP1.4 (Validation).

Figure 1: Aerial view of the studied site in Northern Spain (WRD)

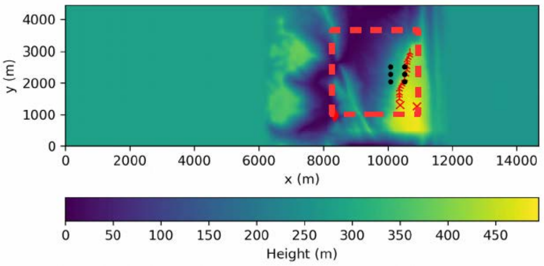


Figure 2: Final model domain with nesting area (red line) and wind turbine positions (red crosses) (IMUK)

Along with an extensive code-to-code benchmark and validation based on a simplified hill geometry [8], analyses of viscous topography and a fundamental investigation of turbulence injection methods (recycling vs. synthetic turbulence) the objective for this work package was the numerical assessment of a reference site featuring extremely high topological complexity with RANS, DES and LES. The studied site located in Northern Spain depicted in Figure 1 is characterized by a pronounced cliff and a series of wind turbines close to the edge. For the PALM code, important achievements from the industrial point of view are the pre- and post-processing tool chain as well as the successful employment of a nesting method, which permits local grid refinement in areas of interest. The tool chain allows for 1. an automatic wind direction alignment, 2. an inflow and outflow smoothing for complex terrain, 3. a preparation kit for turbulence recycling zones, 4. several workflow and efficiency improvements and finally 5. a monitoring feature for instantaneous wind turbine parameters (e.g. wind shear, flow inclination). On the OpenFOAM side, CFDB successfully demonstrated a volumetric synthetic turbulence generation method for complex terrain. Simulations were done for a pre-defined wind direction of 240° (WSW) derived by WRD from a preceding on-site met mast campaign.

IMUK did a series of sensitivity studies to thoroughly eliminate numerical errors. For the development of turbulence and the wind direction near the ground, the influence of the precursor run was investigated. As PALM is a meteorological tool, the wind direction is defined by the geostrophic wind (i.e. pressure gradient force and Coriolis force are in balance) of in this case u_g=15m/s and the velocity near ground must thus be evaluated iteratively. Also the type of the top boundary condition (Neumann vs. Dirichlet) was considered. Of course, the influence of the mesh resolution and the main and nesting domain sizes were looked at. The initial domain size was 23 x 10 x 3 km^3 with a Cartesian grid spacing of 10 m. This particular mesh consisted of $8\cdot10^8$ grid points. Due to the complexity and size of this test case, a domain confinement in combination with the nesting method was thought to yield significant run-time savings. Parameter studies indicated an optimum for a parent domain size of 15 x 5 x 3 km^3, a nesting domain size of 2.5 by 2.5 km^2 around the wind park, a cell size of 10m in the parent domain and 5m in the nesting volume as depicted in Figure 2. The wall clock time on 1000 CPUs could be reduced by a factor of 0.45 from 297 hours to 136 hours comparing the finest grid computations (5m) with the nested runs.

CFDB used snappyHexMesh for grid generation. OpenFOAM's ability to process unstructured grids automatically implies grid refinement capabilities. Figure 3 shows the target grid sizes used for this study. An unavoidable uncertainty stems from the difference in the formulation of the inflow velocity profile between PALM as a meteorological solver and other technically-driven flow solvers like OpenFOAM. As mentioned before, the meteorological situation is described by the geostrophic wind, which causes a wind direction change of up to 17° from the top of the domain to the wind turbine altitude in this case. In the OpenFOAM simulation, the incoming velocity profile is predicted by the underlying RANS part of the hybrid model, and is thus strongly dependent on the exact RANS modelling details. This is less pronounced in the PALM simulation, where the flow profile is predicted by an LES model. Investigations revealed that reaching a perfect conformity of the local velocity distribution between both approaches was not possible within the AssiSt project. It was thus decided to continue with an exact imitation of the geostrophic wind specification at both sides. For the generation of turbulence at the inlet to the scale-resolved portion of the domain, CFDB applied the volumetric synthetic turbulence generation (VSTG) method of Shur [9], which was injected within a length of 3km upwind of the wind park. The hybrid numerical method applied distinguishes between areas of fine grid / turbulent flow and coarse grid / irrotational flow and applies a low dissipative central difference scheme and an upwind scheme, respectively.

Both partners independently observed an influence of the vortex shedding from the rim of the upstream ridge. Strong vortical structures move across the valley located directly in front of the wind park and continually increase the local turbulence level. Additionally, the flow is accelerated at the edge of the wind park plateau leading to a strong flow inclination and wind shear. Figure 4 shows an example history of vertical wind shear level over the horizontal wind speed at wind turbine #8. The red lines indicate certification limits. This wind turbine is potentially prone to observe strong vertical shear.

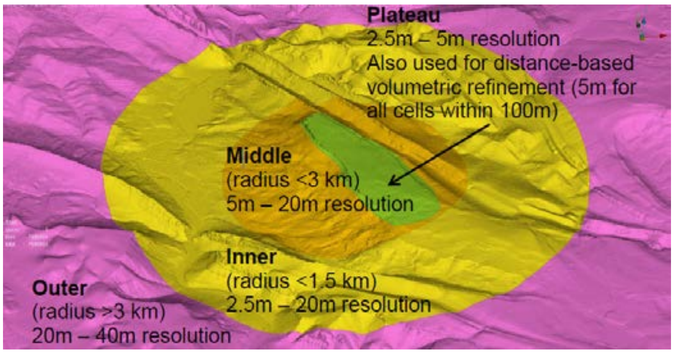


Figure 3: Grid resolution areas for complex site simulation with OpenFOAM (CFDB)

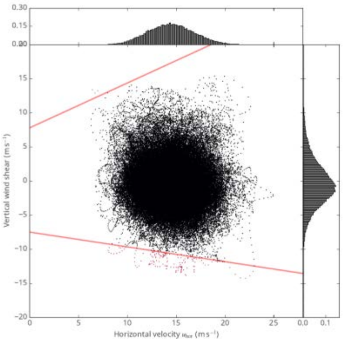


Figure 4: Vertical shear of wind turbine #8 over horizontal wind speed (IMUK)

3.2. *Results of Work Package 2*

Work package 2 was subdivided into 4 Tasks: WP2.1 (Atmospheric Inflow), WP2.2 (Extreme Wind), and WP2.3 (Validation).

Work package 2 relates to the wind turbine itself and establishes a connection between the macroscopic atmospheric situation from work package 1 and the local flow pattern near a selected wind turbine. The hypothesis was that certain flow features resulting from the complex terrain can only be accounted for by incorporating the terrain into the simulation while turbulence statistics gained from meteorological simulations can easily be adopted by a suitable boundary condition. The time-resolved URANS-simulations of a stand-alone wind turbine within complex terrain and with turbulent inflow were done by IAG (FLOWer) and DLR (THETA) and should demonstrate the benefit for forensic yield and load estimation.

A prerequisite for the success of this task was the elaborate investigation of the spatial development of inflow turbulence. IAG looked at the turbulence decay based on a Mann model and a 5th order WENO scheme. The energy spectra delivered a good estimate of the necessary grid resolution. For the overall simulation of the WP1 site, a turbine was chosen that showed the most severe shear levels in the IMUK simulations. The model domain was reduced to the near field of 500m upstream, 400m width and 300m wake based on preliminary studies. The background grid, Figure 5, benefited from the hanging grid node principle with cell sizes of approximately 1m in the near field. The wind turbine was meshed separately and was introduced via overset meshing. The overall mesh consisted of $90 \cdot 10^6$ hexahedral cells. Turbulence was fed in by a newly developed PALM-FLOWer interface.

The approaches of IAG and DLR are quite similar. Especially the grid resolution was adopted from IAG and IMUK. Two grids were considered, one with $42{\cdot}10^6$ and another one with $25{\cdot}10^6$ grid points. THETA required mirroring of one side plane to account for periodicity. Again, turbulence data was induced by a PALM-THETA interface at the same position as IAG.

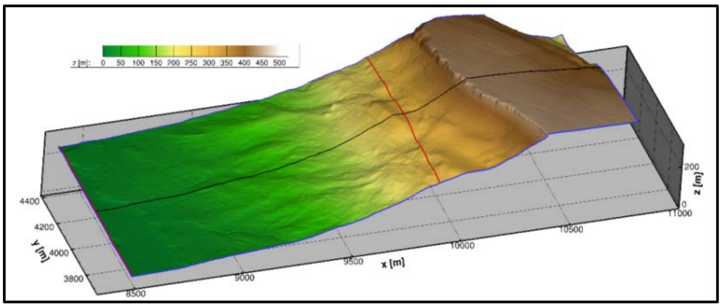


Figure 5: Surface mesh coloured by altitude for the wind turbine simulation in complex terrain (IAG)

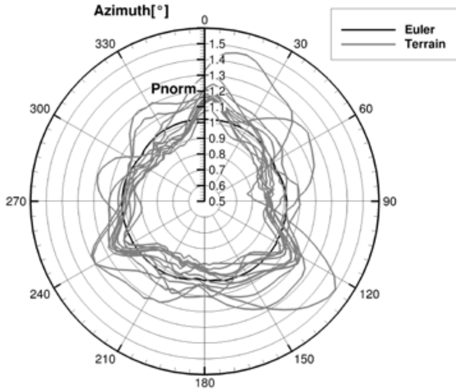


Figure 6: Azimuthal distribution of the integral power with and without terrain (IAG)

Both partners did simulations of the isolated terrain. A common finding was a local separation bubble closely behind the escarpment leading to negative shear at the wind turbine position. IAG found that the adverse pressure gradient due to inclination of the plateau interacts with the induced pressure gradient from the shear layer and momentum extraction of the wake which leads to a stable flow separation and an elliptical deformation and deflection of the wake in the vertical direction. Monitored blade forces, thrust and power were used to highlight the impact of the terrain and atmospheric inflow on the turbine. Figure 6 shows a representative time script of the normalized power over rotor azimuth position. In comparison to a flat terrain benchmark computation (indicated by a black line), the power evolution in complex terrain strongly scatters and emphasizes the limitations of common, idealized “laboratory” conditions.

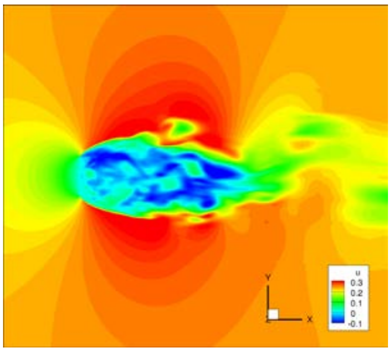


Figure 7: Velocity field of a simplified blade exposed to 90° AoA inflow computed with DES (WRD)

The activities in the “Extreme Wind” work package shed light on the blade aerodynamics with the WEC exposed to extreme wind speed. It was shown that the physical phenomena of high angle of attack aerodynamics exceeded the limits of classical RANS and URANS methods, whereas Detached Eddy Simulation (DES), as exemplarily depicted in Figure 7, mostly alleviated the shortcomings of the RANS methods. It was found that the near wake region is predicted to be excessively coherent by URANS leading to a strong pressure minimum. The integral drag force is thus exaggerated. DES on the other hand predicts finer-scale incoherent vortices at the leading and trailing edge of the profile, thereby reducing the pressure minimum at the surface. Comparisons with experimental forces reveal a good consistency of the DES results. In contrast to an expensive DES, empirical methods also produced satisfying results with only a fraction of the effort.

3.3. Results of Work Package 3

Work package 3 was subdivided into 4 Tasks: WP3.1 (THETA on the Isolated Rotor), WP3.2 (Transition Prediction), and WP3.3 (Validation).

WRD successfully works with the DLR FLOWer code on a research and development level. However, FLOWer as a compressible flow solver exhibits a reduced convergence rate for flow situations at low inflow speed with low compressibility. Thus, the incompressible RANS solver THETA was comprehensively studied for wind energy converters during the project. It could be seen that THETA can successfully be applied to rotor flows at high computational cost reduction rates. Special interest was on the prediction of the laminar-turbulent transition prediction. Figure 8 shows the predicted transition line on the suction side of the generic rotor blade created with a set of public airfoils at an arbitrary pitch and power setting along with a spanwise distribution of the local Reynolds number and the instability indicators for Tollmien-Schlichting and laminar separation transition.

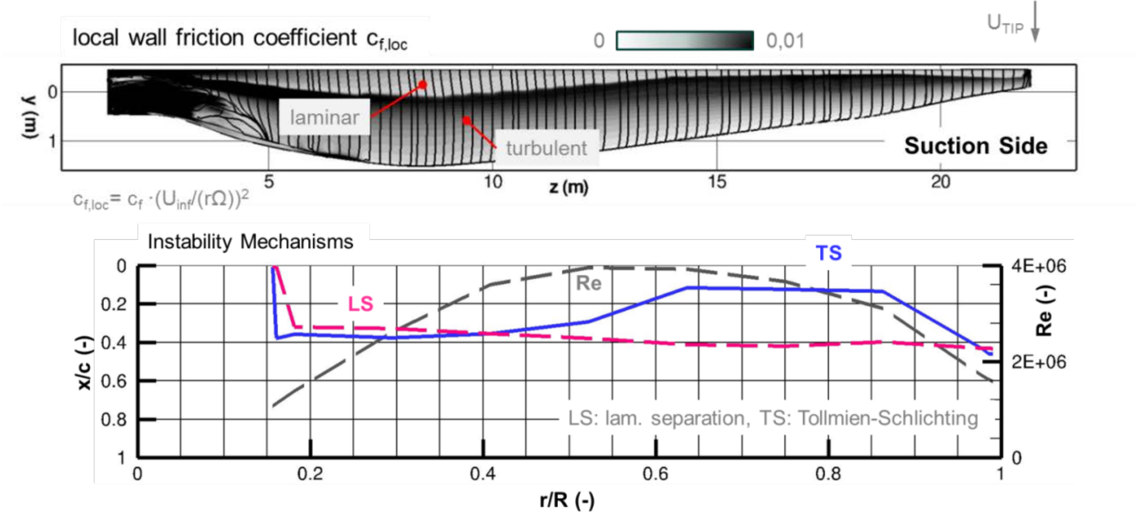


Figure 8: Laminar-turbulent transition position and corresponding Reynolds number and instability mechanism on generic rotor blade at Pitch=2°, Lambda=6, v_{Tip}=60m/s, Tu=0,07% (DLR)

IAG worked on a more robust transition module in FLOWer. The e^N envelope method depends on the integral boundary layer parameters and therefore requires a reliable detection of the boundary layer edge and stagnation point. Modifications on the search algorithm were implemented and successfully demonstrated for this purpose. Figure 9 shows one of the many comparisons between different implemented boundary layer approximations and transport equations based transition methods, in this case on the DU00-W-212 at a $Re=3\cdot10^6$ and an angle of attack of 0°.

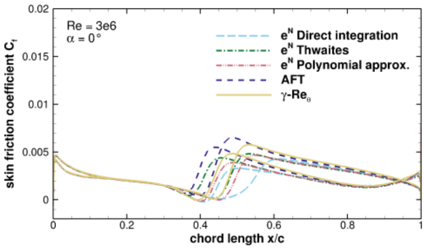


Figure 9: Comparison of different transition prediction methods for a DU00-W- 212 airfoil based on the skin friction coefficient (IAG)

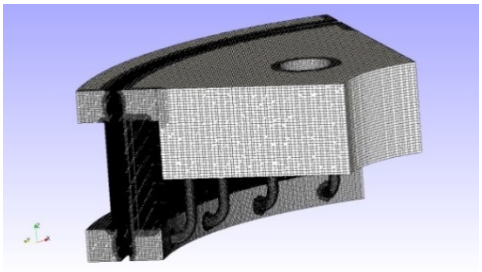


Figure 10: Mesh of a generic generator segment (CFDB)

3.4. Results of Work Package 4

Work package 4 was subdivided into 4 Tasks: WP4.1 (Internal Aerodynamics), WP4.2 (Passive Flow Control), WP4.3 (Validation), WP4.4 (Hub Flow Analysis) and WP4.5 (Hub Optimization).

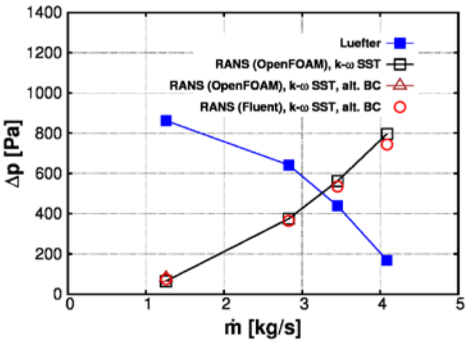


Figure 11: Results of the blind study on the pressure loss in the generic generator segment (WRD, CFDB)

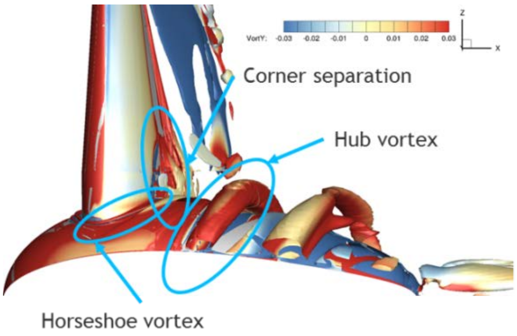
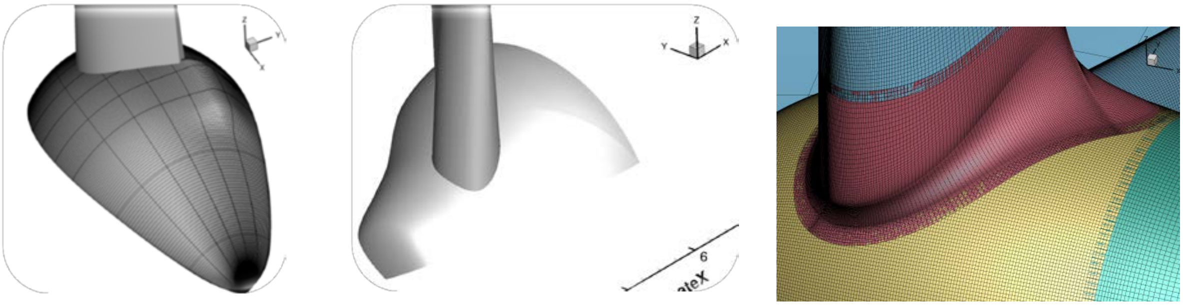


Figure 12: Vortex topology of the flow around the ellipsoidal hub (IAG)

In work package 4, three different system components of the wind turbine were treated in detail. First of all the air cooling of the generator was investigated. The computation of the total pressure loss due to complex flow obstructions was most of all a matter of robust grid generation. OpenFOAM's SnappyHexMesh proved to be a convenient tool to repetitively generate sound grids, which was demonstrated by CFDB for a test stand with coil windings and a generic generator segment, the latter shown in Figure 10. WRD carried out blind studies on both test cases and provided validation data for the test stand. In the end, CFDB and WRD received similar results as depicted in Figure 11, where the maximum difference between the predicted pressure losses was less than 7%. Forensic studies on detailed flow features such as the passing of the coils demonstrated consistency between the individual computations.



(a) Hub plateau (b) Beluga nose (c) Root fairing

Figure 13: Potential hub improvements (IAG)

Secondly, the flow around the characteristic ENERCON hub was explored in depth both in terms of an in-depth analysis of the status quo and in terms of optimization approaches. The interaction of different vortices as depicted in Figure 12 is a fairly complex flow topological situation. IAG carried out simulations with FLOWer and examined the vortex system near the blade root. In front of the leading edge, the computation revealed a horse shoe vortex that was influenced by the pressure gradient, in turn driven by the hub curvature. This vortex united with a corner separation bubble in the wake of the trailing edge segment. The rotational hub vortex built the back end of the vortex system. Two starting points were identified for optimization. The corner separation should be diminished as much as possible and the hub wake ought to be reduced. It was shown that the treatment of the aft portion of the hub as a non-rotating component, which resembles reality, altered the momentum of the hub vortex. This in turn had a positive effect on the hub wake. Concerning the corner separation reduction, the work was accompanied by studies on the effect of turbulence models and numerical settings. Different technological improvements had been taken into account (a subset shown in Figure 13), from which some, like the hub dent or the beluga nose, were designed to relocate the trailing edge of the blade root in a region with favorable pressure gradient. Other ideas like the root fairing directly addressed the corner separation. The latter strategy proved to be more successful.

The work package was concluded with an investigation of vortex generators and airfoils in deep stall. The industrial emphasis in this context was 1. the handling of vortical structures by automatic grid adaptation and 2. the treatment of highly unsteady numerical results in an automated way, which was demonstrated by use of specialized statistical algorithms [10]. This task was carried out by CFDB using OpenFOAM with some customizations. The ability to automatically refine and coarsen the grid in presence of vortex cores, as depicted in Figure 14, simplifies the initial mesh grid generation and alleviates potential user-inflicted shortcomings. The same applies for the automatic detection of statistical convergence from scale-resolved time series. Figure 15 shows the wake of a profile in deep stall computed with Delayed Detached Eddy Simulation (DDES) and characterized by the large number of vortical structures. Low-frequency modulation of the wake flow led to a slow statistical convergence of the time-averaged flow, meaning that long time series must be computed. CFDB's statistical analysis method could be used to derive a 95% confidence interval to automatically determine the right level of convergence, thereby preventing erroneous design conclusions and minimizing user effort.

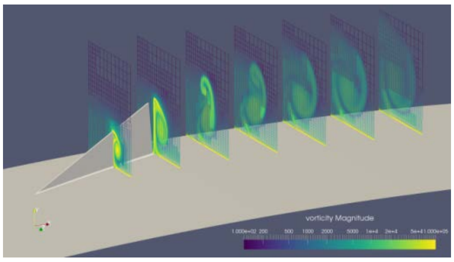


Figure 14: Forming of a trailing vortex behind a single VG with adaptive meshing (CFDB)

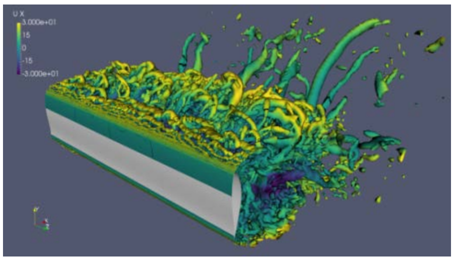


Figure 15: Wake of a profile in deep stall (CFDB)

4. Conclusion

AssiSt was initiated and planned by WRD and its contents reflect its industrial needs in 2013. The project was arranged in a way that the partners worked on generic geometries that closely mimicked their real life counterparts. WRD transferred the developments and findings to its product line uncovering flaws and exploitation potential. The results of the project underlined the value of CFD in the industrial development process. Numerical methods can successfully complement experiments and give valuable insight to inaccessible flow regions (transition, generator cooling, hub aerodynamics). In addition, CFD is a convenient instrument to accelerate component optimization (hub-root optimization, park layout). Nonetheless, the results reveal further potential for certain numerical features such as scale resolved methods (site assessment, deep stall) or multi-physics (aerodynamics, aeroacoustics, structural mechanics, heat transfer). Both, practically and scientifically, the Assist project provided enough material for future research and industrial utilization. Since it is impossible to encompass all scientific findings of the project in this article, the authors would like to refer the interested reader to the following publications covering the topics site assessment of complex terrain [11], [12], [13], wind turbine aerodynamics [14][15][16][17] and internal aerodynamics [18].

The key findings of the project can be summarized as follows:

- Besides a pre- and post-processing tool chain for wind turbine specific analyses, a nesting method was implemented within the LES solver PALM followed by a thorough parameter study for an optimal numerical case setup
- An OpenFOAM based volumetric synthetic turbulence generation method was successfully demonstrated for the complex site case
- Both methods showed an influence of the vortex shedding on the wind turbines leading to increased local turbulence and negative vertical shear events with strong potential impact on the wind park
- URANS simulations were carried out by two partners in the near field of a complete wind turbine in complex terrain with turbulence data gained from the LES simulations in WP1
- Simulations in the isolated terrain delivered local flow separation upstream of the WEC. The interaction of the complex terrain with the turbine wake delivered valuable information about the wake deformation and deflection
- The activities emphasized the limitations of common, idealized inflow conditions and terrain simplifications
- Differences between RANS and scale resolving methods were demonstrated for profiles at very high angles of attack. The shortcomings of RANS were caused by a strong accentuation of the pressure minimum in the near wake
- Empirical methods for the assessment of profile data at high angles of attack produced satisfying results with only a fraction of effort
- Modifications on the search algorithm for the boundary layer edge were demonstrated to produce a more stable and reliable detection of laminar-turbulent transition
- Simulations for the internal flow inside an air-cooled generator segment were carried out. Validation and verification studies showed good consistency between the different solver setups
- A profound analysis of the vortex topology around an ellipsoidal hub was carried out. The complex system of vortical flow structures was elaborated. This information was subsequently used for optimization recommendations on the hub shape
- It was found that a root fairing was the most promising method to successfully eliminate the corner separation, which was identified to be the driving loss factor
- For the flow around a profile equipped with blade vortex generators and a profile in deep stall, the need for automatic grid refinement and coarsening as well as the detection of statistical convergence was highlighted

Acknowledgments

The project AssiSt (Anlagenströmungssimulation und Standortbewertung) was funded by the German Federal Ministry of Economic Affairs and Energy under the grant number 0325719A.

References

[1] Raasch S and Schröter M 2001 PALM – A large-eddy simulation model performing on massively parallel computers *Meteorol. Z.* **10** 363-72

[2] Abd Razak A, Hagishima A, Ikegaya N and Tanimoto J 2013 Analysis of airflow over building arrays for assessment of urban wind environment *Building and Environment* **59** 56-65 DOI: 10.1016/j.buildenv.2012.08.007

[3] Steinfeld G, Tambke J, Peinke J and Heinemann D 2010 Development of a tool for the study of flow conditions and turbulent loads in offshore wind farms *Proceedings of EWEC 2010* 20.-23.04.2010 Warsaw Poland

[4] Meister K, Lutz T and Krämer E 2010 Consideration of unsteady inflow conditions in wind turbine CFD simulations DEWEK Congress 17.-18.11.2010 Bremen Germany

[5] Lutz T, Meister K and Krämer E 2011 Near wake studies of the MEXICO rotor *Proceedings of EWEC* 14.-17.03.2011 Brussels Belgium

[6] Lambert M and Kessler R 2010 Radiation heat transfer in mixed convection flows *New Results in Numerical and Experimental Fluid Mechanics VII - Notes on Numerical Fluid Mechanics and Multidisciplinary Design* **112** 563-70

[7] Weller H, Tabor G, Jasak H, and Fureby C 1998 A tensorial approach to computational continuum mechanics using object-oriented techniques. *Computers in physics*, **12**(6), 620-631

[8] Ishihara T, Hibi K and Oikawa S 1999 A wind tunnel study of turbulent flow over a three-dimensional steep hill *Journal of Wind Enginnering and Industrial Aerodynamics* **83** 95-107

[9] Shur M, Spalart P, Strelets M and Travin A 2014 Synthetic turbulence generators for RANS-LES interfaces in zonal simulations of aerodynamic and aeroacoustic problems *Flow, Turbulence and Combustion* **93** 63-92. DOI: 10.1007/s10494-014-9534-8

[10] Mockett C, Knacke T and Thiele F 2010 Detection of initial transient and estimation of statistical error in time-resolved turbulent flow data. In *8th International Symposium on Engineering Turbulence Modelling and Measurements (ETMM8), Marseille, France*

[11] Böske L N, Knigge C and Raasch S, 2018 High-resolution LES for site assessment in complex terrain *to be published in 2018*

[12] Hellsten A, Ketelsen K, Maronga B, Sühring M, Knigge C, Barmpas F, Tsegas G, Auvinen M, Moussiopoulos N and Raasch S 2018 A nested multi-scale system implemented in the LES model PALM *to be published in 2018*

[13] Knigge C, Gronemeier T, Inagaki A and Raasch S 2018 Comparison and improvement of two different methods to generate turbulent inflow data for LES *to be published in 2018*

[14] Weihing P, Letzgus P, Bangga G, Lutz T and Krämer E 2018 Hybrid RANS/LES capabilities of the flow solver FLOWer – application to flow around wind turbines *Preprint Notes on Numerical Fluid Mechanics and Multidisciplinary Design*

[15] Schulz C, Letzgus P, Weihing P, Lutz T and Krämer E 2018 Numerical simulation of the impact of atmospheric turbulence on a wind turbine in complex terrain in *to be published in Proceedings of The Science of Making Torque from Wind* 20.-22.06.2018, Milano Italy

[16] Weihing P, Wegmann T, Lutz T and Krämer E 2018 Numerical Analyses and Optimizations of the Flow in the Nacelle Region of a Wind Turbine *to be published in Wind Energy Science*

[17] Länger-Möller A 2018 Investigating the impact of wall roughness and turbulence level on the performance of a horizontal axis wind turbine using CFD *to be published in Wind Energy*

[18] Mockett C, Fuchs M, Knacke T, Kramer F, Kühn T, Altmikus A, Thiele F 2017 Assessment and enhancement of OpenFOAM for various wind engineering applications 5th OpenFOAM User Conference 17.-19.10.2017 Wiesbaden Germany