Coupled Fluid-Structure Simulations of a Wind Turbine Rotor

Swen Streiner[1,2], Stefan Hauptmann[2], Martin Kühn[2], Ewald Krämer[2]
[1]Aero Dynamik Consult GmbH, Stuttgart, +49(0)7158 956551-40, streiner@aero-dynamik.de
[2]Universität Stuttgart, Germany, +49(0)711 685 60339, hauptmannn@ifb.uni-stuttgart.de,
kuehn@ifb.uni-stuttgart.de, kraemer@iag.uni-stuttgart.de

Summary

This paper presents an approach to compute fluid-structure interactions on wind turbines. It is a contribution to the development of future design tools and aims to improve the quality of numerical simulations of the fluid-structure interaction process, leading to a better understanding of the underlying physics. The presented approach is widely discussed in literature and is referred to as tight or strong coupling.

Strong coupling means an exchange of fluid loads and structural deformations at each time step. Since the analysis methods and codes for both domains have independently reached a high level of sophistication, this approach is effectuated in a fully modular manner and data is exchanged between separate codes. The underlying coupling schemes are classified by the character of time integration on fluid and structure side, respectively. Several combinations are possible, but this paper focuses on a first order implicit-explicit scheme.

So far the strong coupling focuses on rotor only computations. The respective models on both fluid and structure side are presented and discussed. The contribution presents coupled fluid-structure computations at the rotor of a 2.75 MW wind turbine. The results are compared to and validated against state of the art simulation tools.

1. Introduction

The dynamics and response of a wind turbine during operation depends significantly on the total damping of the turbine, which is a combination of the structural and aerodynamic damping. Usually, the structural contribution to the damping is well known while the aerodynamic part can be very uncertain. Aerodynamic damping depends upon the operational conditions, e.g., wind speed and actual rotational speed and blade pitch. In some cases, the aerodynamic part of the total damping can be negative, resulting in severe vibrations that eventually can lead to structural failures of wind turbines. Thus, the need for developing prediction tools to avoid such stability problems is obvious.

For more detailed insights in the dynamic behaviour of wind turbines, improvements of aerodynamic as well as interrelated structural dynamic computations are needed. Currently the increase of computing power and better insight in the physical behaviour of wind turbines are bringing new powerful simulation techniques to the wind energy community. On the one hand, first applications of computational fluid dynamics (CFD) methods are approaching industrial applications. On the other hand, structural failures in the mechanical systems drive the interest for more accurate modelling of the wind turbine dynamics e.g. by multibody simulations. A key for a breakthrough in wind turbine technology is to reduce the uncertainties related to blade dynamics, by the improvement of the quality of numerical simulations of the fluid-structure interaction process, and by a better understanding of the underlying physics.

This paper links both developments and presents an approach to fully coupled fluid-structure computations based on computational fluid dynamics (CFD) and multibody simulation (MBS). More realistic wind turbine simulations are, however, a multi disciplinary problem and only first results of the ongoing efforts will be presented.

The chosen procedure is to couple separate methods for aerodynamics and structural dynamics by a simply staggered ”Partitioned Procedure” approach. In this way the continuous, physical interaction is replaced by discrete influence terms. The conservation laws of both domains are solved separately, which has the advantage that well established solution methods can be used, and code alterations are limited to a minimum. For the present study an implicit-explicit staggered scheme, which is iteration free and of first order time wise accuracy is used.

2. Aerodynamics

FLOWer [1] has been developed to solve the three-dimensional, unsteady Euler or Reynolds averaged Navier-Stokes (RANS) equations in order to analyse the flow field around the helicopter rotor. The equations are formulated in a hub attached, non-inertial rotating frame of reference with explicit contributions of centrifugal and coriolis forces. The discretization of space and time is separated by the method of lines using a cell-vertex finite volume formulation. Spurious oscillations of the central difference scheme are suppressed by first and second order artificial dissipation. The time integration makes use of the dual time stepping technique with a second order implicit time integration operator. Different turbulence models are available in FLOWer. However, due to good experiences in former studies the k-ω SST turbulence model [2] is the sole model used for the present study.

FLOWer features the Chimera technique [1] allowing for arbitrary relative motion of aerodynamic bodies. Body fitted refined grids around each blade are embedded in a background grid, in which the blade vortex sheets are convected from one blade grid to the next. The elastic deformation of the blade can be introduced into the body fixed multiblock grids by an algebraic deformation method for OH-grid and CH-grid topologies.

Uncoupled unsteady RANS (URANS) rotor only computations have been preformed and validated for the considered 2.75 MW wind turbine in previous studies, e.g. [3]. In order to achieve reliable results for the pressure distribution on the blade surface, a highly resolved grid is mandatory. Grid quality with respect to boundary layer resolution is verified by examination of the dimensionless wall distance value y^+, where values of about one should be aimed for. Due to changing flow conditions along a rotor blade and in order to limit the grid to a sensible size most often a good compromise has to be found in this respect. The following Figure 1 shows the y^+-distribution for the applied grid as a measure for the very good grid quality used in this study.

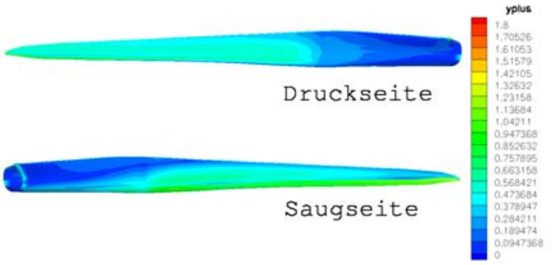

Fig. 1: y^+-distribution at u=10m/s at pressure and suction side.

The overall setup consists of the main grid components blade, hub and background grid, leading to total number of about 10 million grid points. An insight to the grid setup is given in the following Figure 2.

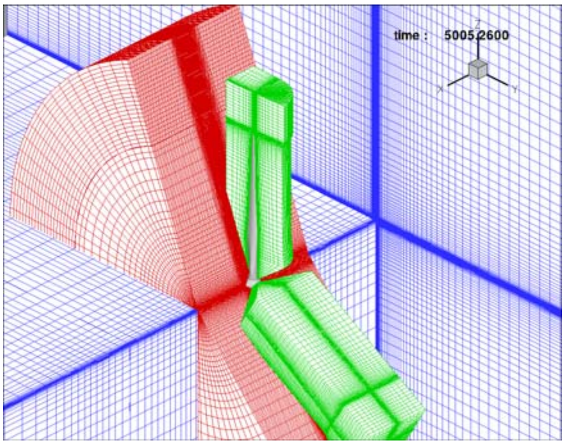

Fig. 2: Grid setup showing blade (green), hub (red) and background grid (blue).

A suitable timestep size for numerical analysis of wind turbine rotors is case dependent and found by conducting 'timestep-convergence studies'. A usually sufficient timestep size for rotor only RANS computations corresponds to an azimuth movement of $\Delta\Psi = 1°$ per timestep. For the coupled computations of this paper a timestep size of $\Delta\Psi = 0.5°$ is used, which is equivalent to a $\Delta t = 4{,}8 \cdot 10^{-3}$s.

3. Structure dynamics

For the structural dynamic computations (CSD) the multibody simulation software SIMPACK is used. SIMPACK is a general purpose three dimensional multi-body simulation (MBS) software tool which is used to aid the development of any mechanical or mechatronic systems, ranging from single components through to complete systems (e.g. wind turbines, vehicles, and combustion engines). The basic concept is to create a CAD style MBS model of a system and to analyse the static and dynamic behaviour of the system. SIMPACK also offers the opportunity to implement flexible bodies to a MBS model using the modal reduction of finite element models. A tool to create flexible rotor blades for wind turbines, based on Euler-Bernoulli beam elements, is also available in SIMPACK.

For the rotor only coupled computations in this paper, the rotor is currently modelled as a stiff hub with flexible rotor blades based on Euler-Bernoulli beam elements.

4. Fluid-structure coupling

Since in both domains, i.e. in CFD and CSD, the analysis methods have independently reached a high level of sophistication and an optimum degree of problem specific adaptation, it would be unreasonable not to exploit the respective expertise. Consequently, preference is given to a modular approach for tackling the fluid-structure interaction problem.

A staggered algorithm approach where exchange of aerodynamic loads and structural deformations takes place at each timestep is generally named strong coupling. The approach is fully modular, since data is exchanged between separate codes. The developed coupling routine is written in the C programming language accessing standard socket connection routines and therefore exchanging data via TCP/IP interface. The coupling routine is platform independent, transforming the data from host format to network format and back again on the client system. The underlying coupling schemes are classified by the type and order of the integration scheme used for the fluid and the structure, respectively. Several combinations are possible, but this paper focuses on a first order implicit-explicit scheme, due to the prescribed integrations schemes of the two codes. The accuracy has been shown in terms of energy exchanged at the common boundary by a mathematically rigorous proof in [4].

The currently used implicit-explicit coupling scheme basically consists of the following two steps, also shown in Fig. 3:

1. Explicit step: computation of structure state at time step n+1 based on current state of fluid and structure.

2. Implicit step: computation of fluid state at time step n+1 based on state of fluid at time step n and state of structure at time step n+1.

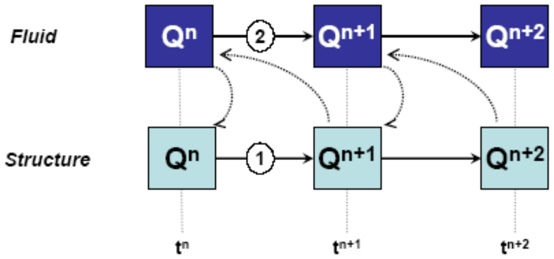

Fig. 3: First order implicit-explicit coupling scheme.

At each time step aerodynamic forces and structural kinematics are exchanged at nodal positions of the rotor blades. In order to adapt the aerodynamic loads to the coarser structure model, the principle of virtual displacements is applied to redistribute the forces to the structure model. The displacements provided by the structural dynamic computation is so far linearly interpolated to the quarter chord nodal positions of the finer CFD grid.

The fully coupled computations require the computational mesh to deform with the geometry. Therefore the CFD computations are performed using highly-resolved deformable chimera grids for the blades which are automatically adapted based on the respective nodal point movements obtained from the structural dynamic computations with SIMPACK.

5. Results

This paper demonstrates coupled fluid-structure computations at the rotor of a 2.75 MW wind turbine. The results are compared to SIMPACK computations connected to the Blade Element Momentum (BEM) code Aerodyn. This method has been validated in previous projects, therefore presenting a solid basis for comparison [5]. In the following Figures 4 to 8, the red lines (dashed) refer to the coupled CFD-computations and compare rotor thrust, rotor torque and blade tip kinematics.

In general the coupled FLOWer-SIMPACK computations show the same trend as those obtained by the SIMPACK-Aerodyn analysis. However, since the simulation time is rather short for the FLOWer-SIMPACK computations, which additionally are preformed at a considerably smaller timestep, a reliable prediction with respect to a final convergence of both computations is not possible on basis of the available data.

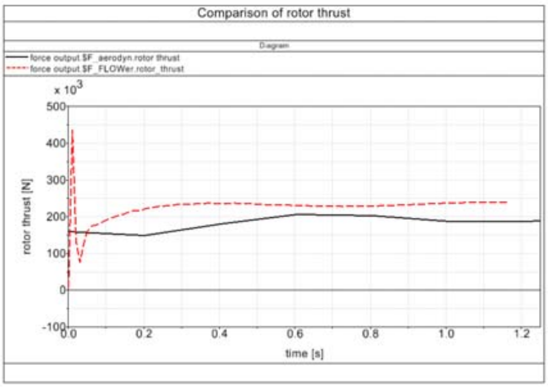

Fig. 4: Comparison of rotor thrust (--- CFD, — BEM).

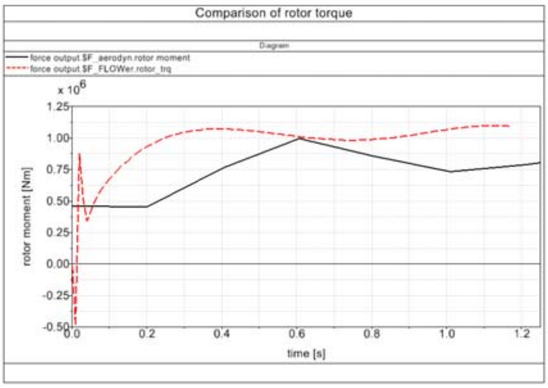

Fig. 5: Comparison of rotor torque. (--- CFD, — BEM)

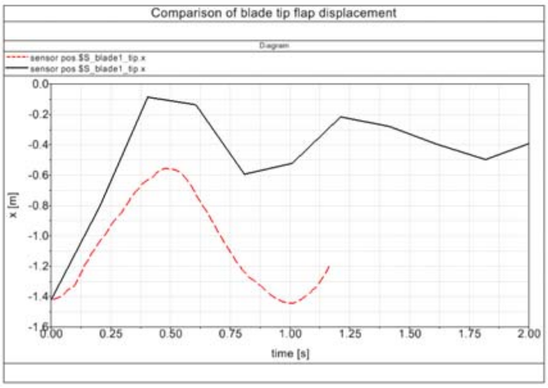

Fig. 6: Comparison of blade tip displacement in flapwise direction. (--- CFD, — BEM)

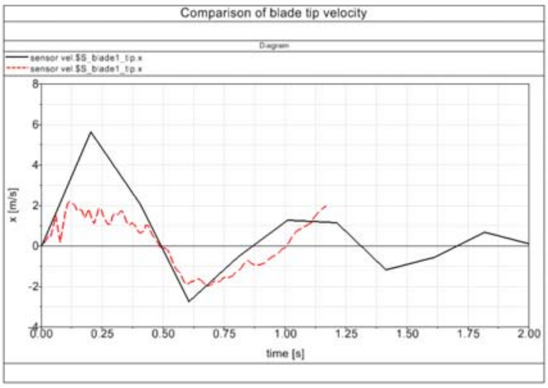

Fig. 7: Comparison of blade tip velocity in flapwise direction. (--- CFD, — BEM)

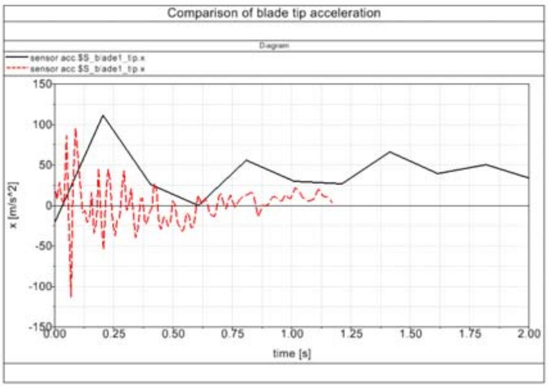

Fig. 8: Comparison of flapwise acceleration at blade tip. (--- CFD, — BEM)

In Figure 9 the thrust coefficient of the first blade is drawn over the blade radius at the last timestep of the respective analysis. Here the CFD coupled computations show a different spanwise distribution, with higher values near the blade root section followed by a steady decrease towards the blade tip (bottom left diagram in Fig. 9). This difference does not seem to be due to numerical instabilities, since the CFD computations show a good convergence (compare Fig. 10) and should also consider the effects of 3D flow over the blade more precisely.

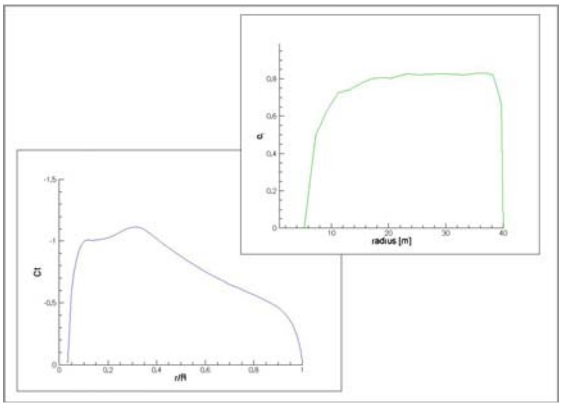

Fig. 9: Comparison of thrust coefficient along blade 1. (left: CFD; right: BEM)

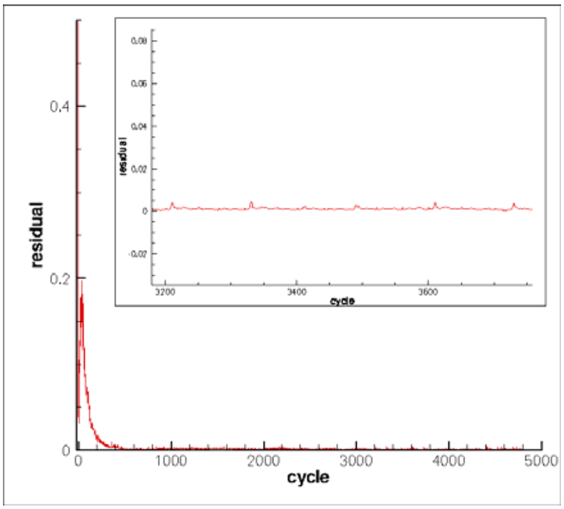

Fig. 10: Convergence of CFD computations

6. Conclusions

An approach for strong coupled fluid-structure simulations between a CFD solver and a MBS code has been implemented and shown to work properly. Since CFD computations are rather expensive with respect to CPU time, a first comparison to standard BEM coupled analyses has been limited to a short time period and needs to be expanded once the functionality of the coupling is fully validated. Although first results are promising, more extensive computations are needed for reliable validation.

In the next steps, the detailed structural dynamics of a complete wind turbine will be taken into account by the SIMPACK model and CFD computations will be extended to a complete wind turbine as well, in order to account for the tower influence. Additionally coupling scheme refinements and improvements in force and displacement exchange are anticipated. With a suitably refined and validated fluid structure interaction approach, computations including turbulent inflow in case of the CFD computations are aimed for.

References

[1] *"FLOWer: Installation and User Manual of FLOWer MAIN version FLOWer 1-2007.1."* DLR Braunschweig, 2007. – Release 2007.1

[2] Menter, F.R.: *"Zonal Two Equation k-ω Turbulence Models for Aerodynamic Flows."* AIAA paper (1993)

[3] Streiner, S. and Krämer, E.: *"Aeroelastic Analysis of Wind Turbines Applying 3D CFD Computational Results".* 2007 J. Phys.: Conf. Ser. 75

[4] Altmikus, A.R.M. and Wagner, S.: *"On the Timewise Accuracy of Staggered Aeroelastic Simulations of Rotary Wings."* AHS Aerodynamics, Acoustics, and Test and Evaluation Specialist Meeting, San Francisco, CA (2002).

[5] Hauptmann, S. et al.: *"Aero-elastic Simulation of a Wind Turbine and Drive Train Resonance Analysis Using the Multi-Body Simulation Code SIMPACK"* DEWEK, Bremen, Nov 2006