CFD studies of a 10 MW wind turbine equipped with active trailing edge flaps

E. Jost[1], A. Fischer[1], Th. Lutz[1], E. Krämer[1]
[1]University of Stuttgart, Institute of Aerodynamics and Gas Dynamics (IAG),
e.jost@iag.uni-stuttgart.de

ABSTRACT

The active trailing edge flap concept is a very promising approach to reduce the dynamic loading of large wind turbines. Over the last years it has been widely researched with different methods as e. g. engineering models making use of the blade element method. The aim of the authors' work is to further analyse the concept in a high fidelity Computational Fluid Dynamics (CFD) environment. Therefore, this article gives an overview on different possibilities to realise actively deflecting flaps in CFD. Three different concepts, one based on the CHIMERA technique and two based on grid deformation, have been examined in 2D and a numerical setup for 3D simulations has been investigated.

1. Introduction

One of the main focus points of today's wind energy research is the accurate determination and reduction of the turbine loads. This is caused by the ambition to realise bigger turbines with higher power output to achieve a lower cost of energy. Many current research projects concentrate on the development of 10 MW turbines. One example is the european-funded INNWIND.EU project, which examines the generic DTU 10 MW reference wind turbine [1]. The realisation of these large turbines is very challenging since simple up-scaling is hardly possible. As described in [2], power, thrust, aerodynamic and centrifugal forces as well as the rotor mass rise disproportionately with the rotor radius. Applied on current market turbines, up-scaling will lead to big, heavy weight rotors which is undesirable because of many economic and technical reasons. It is therefore necessary to develop new concepts in the aerodynamic, structural and controller design of these large turbines. One of the main aerodynamic challenges is to reduce the dynamic loading caused by tower shadow, atmospheric boundary layer or turbulent inflow. As shown by several scientists, e.g. Barlas [3], active trailing edge flaps (ATEF) constitute a very promising approach for these matters. The majority of them researched the technology with the use of blade element or free vortex methods. The aim of this work is to investigate the ATEF concept with CFD. Two dimensional simulations of the NACA 64_3618 have been performed to evaluate the different possibilities to enable active flaps in CFD. Additionally to the trailing edge flap (TEF) the airfoil is equipped with a leading edge flap (LEF) to validate both cases. Furthermore three dimensional simulations were conducted on the DTU 10 MW RWT in a 120 degree model. The basic principle behind ATEFs is very simple. Deflecting flaps attached to the rear of a profile are able to influence the lift coefficient for a given angle of attack. A change in the incoming wind speed leads to a variatian of the angle of attack and thereby an alleviation or increase of lift resulting in the turbine loads. Active trailing edge flaps counter this effect with the attempt to keep the lift constant by adapting the flap deflection to the current wind condition.

2. Realisation of active flaps in the CFD code FLOWer

The simulations were performed with the compressible Navier-Stokes solver FLOWer which was developed the German Aerospace Center [4]. FLOWer is a block structured code based on finite volume formulation. The implementation of the CHIMERA technique for overlapping meshes [5] in addition to the Dual-time-stepping method for time-accurate simulations allow flow computations around moving or rotating bodies as e.g. wind turbines. In FLOWer, active devices like flaps can be realized in two different ways, through grid deformation and based on the CHIMERA technique. So far, two different types of grid deformation algorithms are implemented, the first based on hermite polynomials [6] and in a newer code version one based on radial basis functions [7]. In the following they are described in detail and referred to as CHIMERA flap, HP flap (hermite polynomials) and RBF flap (radial basis functions). For simplification the functioning is shown on an airfoil only equipped with TEF. The geometry of wind turbine flaps differs to those commonly used in aviation as the flap is directly hinged to the airfoil and moving in a circle-formed chamfer. There is no gap between the main and the flap part of the airfoil.

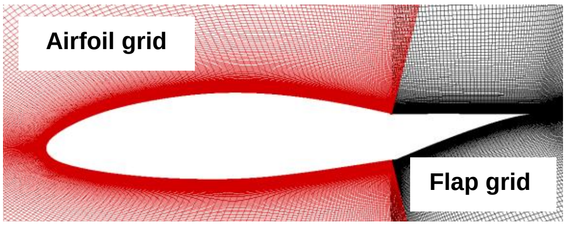

Figure 1: CHIMERA flap

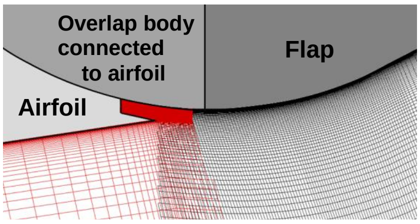

Figure 2: Gap closing CHIMERA flap

The modeling based on the CHIMERA technique requires the use of two completely separate grids for the airfoil and flap part. These grids are overlapped and holes are cutted out of the airfoil grid where the flap is located and respectively to the flap grid where it connects to the airfoil. Figure 1 shows the final computational grid. The small gap between airfoil and flap, that is created through the separation into two different grids, is closed like it is shown in figure 2. This is necessary to avoid flow through the split which can affect simulation results. Figure 2 also shows how the airfoil is extended with an overlap part to achieve an accurate CHIMERA interpolation. A obvious disadvantage of the CHIMERA flap is the elaborate grid generation process already in 2D. In addition, two grids lead to a higher amount of cells and therefore longer computation time.

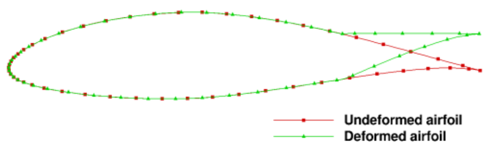

Figure 3: Undeformed and deformed surface

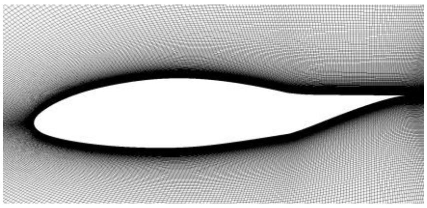
Figure 4: Deformed computation grid

The basic principles of the two possibilities based on grid deformation, the HP flap and the newer RBF flap, are identical. First, the deformed surface is obtained by rotating the flap part of the surface around a defined hinge point. Figure 3 shows the undeformed original surface in addition to the deformed surface with flap deflection. Based on these surfaces the deformation of the simulation grid is computed as illustrated in figure 4. The difference of the two options results from the deformation algorithm and is their variability concerning grid topology. The HP algorithm calculates the deformation of the nodes based on their i,j,k-index of the structured grid. It is therefore necessary to stick to certain restrictions regarding the block structure. In the RBF algorithm the deformation is computed based on x,y,z coordinates and independent of grid topology. Both algorithms needed adaption to achieve accurate results in the transition region from airfoil to flap. In figure 5, the issue is shown examplarily on the RBF algorithm with the lower side of the airfoil displayed. The problem is that the rotation results in a gap in the surface discritization. For good results the local node distribution had to be adjusted by the use of two sided stretching functions [8] like it is shown in figure 5b. The HP version showed the same issue in a similar manner and was therefore also adapted.

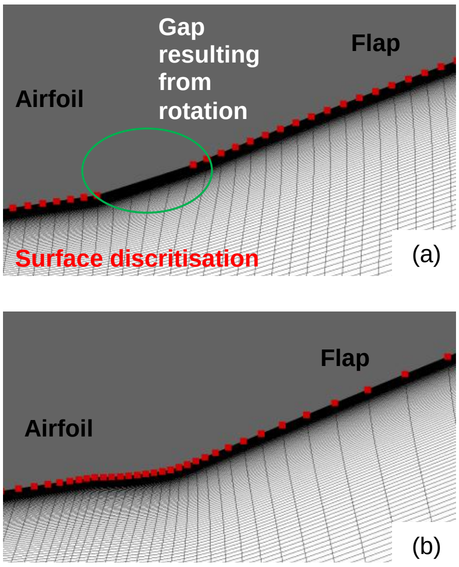

Figure 5: RBF grid before (a) and after (b) redistribution

In general a modeling based on grid deformation is favorable in comparison to the CHIMERA approach. A smaller amount of cells (approx. 130,000 in a 2D case) and grids and no CHIMERA intersection near the airfoils surface are major advantages concerning simulation time and accuracy.

3. 2D results single trailing edge flap case and combined leading/trailing edge flap case

The implementations have been tested in a single TEF case and in a combined LEF/TEF case for which measurement data is available. The data was obtained by Lambie [9] who researched a self-adapting camber concept. The HP algorithm was only investigated in the TEF case as no LEF implementation has been made so far. Up to now the implementation is also not planned because of the major grid topology restrictions that the algorithm needs. The CHIMERA flap and the RBF flap have been simulated for both cases as it is shown in figure 6 and 7.

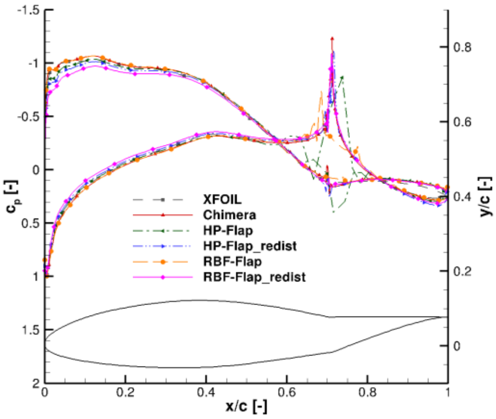

Figure 6: c_p single TEF case (α=8.455°, β_{TEF}=14.63°)

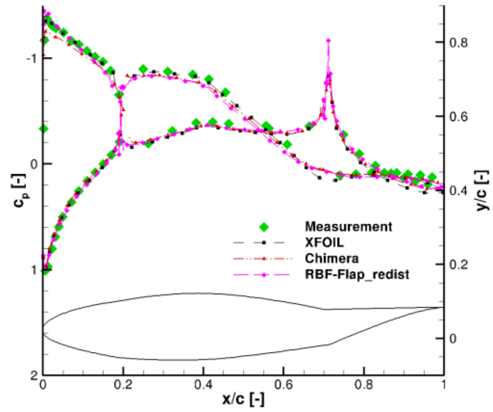

Figure 7: c_p LEF/TEF case (α=9.07°, γ_{LEF}=-5.61°, β_{TEF}=15.98°)

The notation '_redist' indicates the redistributed version of the deformation algorithms. For comparison a xfoil-simulation is also shown. A good accordance can be seen in all cases but the undistributed deformation cases where there are strong fluctuations in the transition areas from airfoil to flap. This is because of the minor grid quality as it has been explained in chapter 3. Further results can be found in [10] and [11].

4. 3D Simulation in a 120 degree model of the DTU 10 MW RWT blade

As the RBF flap offers the needed flexibility regarding grid generation and is easier to handle than the CHIMERA flap with respect to the setup, it has been chosen for the first 3D simulations with static flap deflection. The main objective of the simulation was a first functionality test to examine the numerical setup and the grid deformation in 3D. Since so far the redistribution algorithm is not available in 3D, special focus has been put on the computation grid. Figure 8 shows a TEF deflection of 10 degrees extending from approx. 82 % to 95 % rotor radius with 10 % chord wise length. The gaps between the flap part and the main part of the rotor blade are kept closed to ease this first computation. Another reason is that as the simulation has been made with a timestep corresponding to 2° azimuth, unsteady gap effects cannot be resolved.

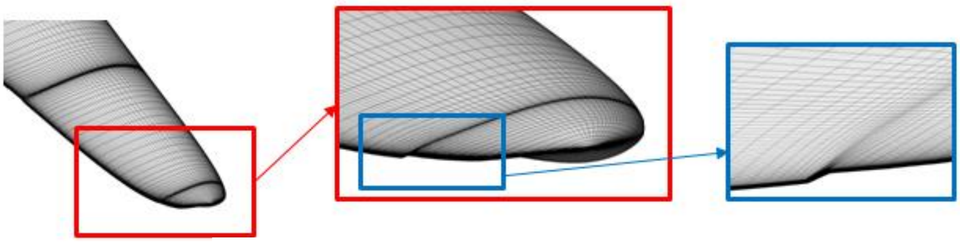

Figure 8: DTU 10 MW RWT 3D surface grid

The computation setup consists of 4 different meshes, background, spinner, nacelle and blade, which sum up to approx. 23.7 million grid cells. In this case the blade is highly resolved with 201 nodes around the airfoil and 261 in spanwise direction to adapt to the flap deflection. As turbulence model the Menter SST was used and to accelerate convergence the MultiGrid method was applied. The simulation was carried out at rated wind speed of 11.4 m/s and 9.6 rpm which leads to a Reynolds number of 6.15 million. In total about 50,000 CPUh were required to achieve a converged solution. In figure 9 results of the simulation are displayed in form of a λ_2 vortex visualization. The blade tip vortex as well as the vortex caused by the change of circulation because of the flap deflection (red circle) can be clearly seen. The flap reduces the aerodynamic power is reduced from 11.02 MW to 10.86 MW and the rotor thrust is increased from 1.77 MN to 1.89 MN. This unintended power reduction is caused by a high drag coefficient resulting from a too large TEF deflection angle. But since the simulation was run with the purpose of a functionality test a large deflection is reasonable.

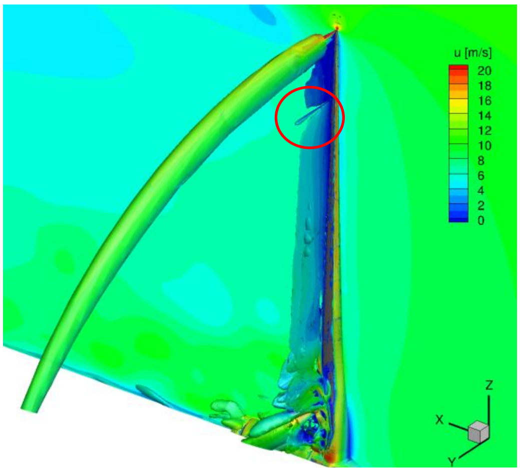

Figure 9: λ_2 vortex visualisation

5. Conclusion and Outlook

This article gives an overview over different possibilities to realise ATEFs in CFD. A modeling based on grid deformation is regarded favorable as less grid cells are needed and a higher accuracy in ensured than with the CHIMERA approach. In oder to determine actual load variations on the examined 10 MW turbine full 3D simulations including tower and atmospheric inflow are planned for the future. As the tower shadow has the most pronounced influence, this will be the first focus point for a simulation with active flaps. Since it is possible to evaluate loads during simulation runtime, control concepts can also be applied. For the beginning the flap deflection will be implemented as function of the blade azimuth.

6. Acknowledgements

For providing their resources the authors gratefully thank the *High Performance Computing Center Stuttgar* and the AVATAR project for funding..

REFERENCES

[1] C. Bak; F. Zahle; R. Bitsche; T. Kim; A. Yde; L.C. Henriksen; P.B. Andersen; A.. Natarajan, M.H. Hansen; “Design and performance of a 10 MW wind turbine”; J. Wind Energy; To be accepted

[2] R. Gasch; J. Twele; “Windkraftanlagen”; Vieweg+Teubner; 7.Auflage ; ISBN 978-3-8348-1460-9; 2011

[3] T. Barlas; “Active aerodynamic load control on wind turbines”; PhD thesis; TU Delft; 2011

[4] N. Kroll; J. Fassbender; “MEGAFLOW – Numerical Flow Simulation for Aircraft Design”; Springer Verlag Berlin/Heidelberg/New York; ISBN 3-540-24383-6; 2002

[5] J.A. Benek; J.L. Steger; F.C. Dougherty; P.G. Buning, Arnold Engineering Development Center; Arnold AFS TN.; “Chimera: A Grid-Embedding Technique”; Defense Technical Information Center; 1986

[6] K.-H. Hierholz; S. Wagner; “Simulation of Fluid-Structure Interaction at the Helicopter Rotor”; ICAS 98 Proceedings; International Council of the Aeronautical Sciences; Stockholm; Sweden; and AIAA; Reston, VA; 1998

[7] M. Schuff; P. Kranzinger; Manuel Keßler; E. Krämer; “Advanced CFD-CSD coupling: Generalized, high performant, radial basis function based volume mesh deformation algorithm for structured, unstructured and overlapping meshes”; 40th European Rotorcraft Forum; Southampton; UK; September 2014

[8] M. Vinokur; “On One-Dimensional Stretching Functions for Finite-Difference Calculations“; Journal of Computational Physics 50; p. 215 – 234; 1983

[9] B. Lambie; “Aeroelastic Investigation of a Wind Turbine Airfoil with Self-Adaptive Camber”; Doctoral thesis; TU Darmstadt; 2011

[10] K. Busse; “Numerische Untersuchung eines Profils mit gekoppelter Vorder- und Hinterkantenklappe unter Verwendung der Chimera-Technik”; Bsc. Thesis; Universität Stuttgart; May 2014

[11] M. Bühler; “Numerische Untersuchung und Optimierung des Modells für Hinter-und Vorderkantenklappen im Strömungslöser FLOWer“; Bsc. Thesis; Universität Stuttgart; October 2014