NUMERICAL FLOW SIMULATION OF A HORIZONTAL AXIS WIND TURBINE (HAWT) ROTOR TO INVESTIGATE THE POTENTIAL OF IMPROVING THE OPERATING PERFORMANCE UNDER USE OF DIFFERENT AERODYNAMIC MODIFICATIONS

Graduate Engineer Sebastian Wittig
cp.max Rotortechnik GmbH & Co. KG, Dresden, Germany

Summary

Appearing flow separation and cross flow at rotor blades of HAWT in the hub area shall be analysed by means of a numerical flow simulation. Based on that, the influence of different aerodynamic modifications (boundary layer fences, flow suction) on the operating performance of the turbine is investigated.

1. Introduction

High bending moments act on rotor blades of HAWT in the hub area. Structural issues in this region compromise the aerodynamic shape of the rotor blades. Circular cross-section areas and thick profiles force massive flow separation. Due to centrifugal forces a secondary flow in radial direction occurs, the so called cross flow [1]. This phenomenon is visualized very well in field tests by means of tufts (Fig. 1).
One possible option for increasing the power performance of operating wind turbines can be found by reducing the flow separation and cross flow by use of different aerodynamic modifications [2]. The results of the flow simulation give a deep insight in the flow phenomena appearing in the hub near area and the influence of different aerodynamic modifications.

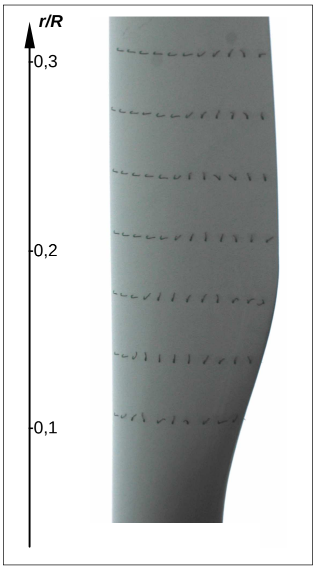

Fig. 1, Field test with tufts in the near hub area on suction side

2. Numerical Simulation

2.1 Numerical Model
The numerical simulation includes a three-dimensional, steady, fully turbulent flow calculation with a 3D-Navier-Stokes flow solver of the program package Fine™/Turbo (Numeca). The one equation turbulence model by Spalart and Allmaras is applied.
To reduce the calculation effort several simplifications had to be applied at the model. The influence of the tower and the atmospheric boundary layer is not considered. Furthermore the rectangular shape of the nacelle is represented by a cylindrical body. This allows treating the wind turbine as a system with rotational symmetry. In that case only the space between two symmetry planes has to be considered in the computations. The resulting structured grid consists of about 4.4 million grid nodes.

2.2 Validation of the Numerical Model
To validate the numerical model results of three different operational points are compared with experimental data. They include measurements on a working turbine and the flow visualisation by the use of tufts in the area of interest. A good qualitative correlation is found between theoretical and experimental results (Fig. 2).

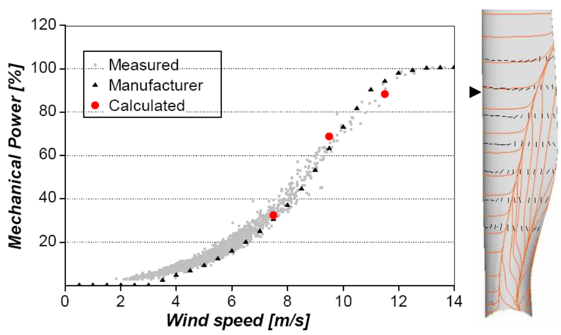

Fig. 2, Validation of the calculation results by means of measurements (left) and tufts (right, black lines indicate tufts orientation, red lines indicate computed streamlines on the rotor surface)

3. Modifications

For one operational point the created model is enhanced by several modifications (Fig. 3). The effect on the flow field, on the flow separation and on the power output is investigated.

Manipulation of cross flow by boundary layer fences of different dimensions and a cylindrical segregation plane is the centre of interest. Moreover the influence of boundary layer suction and a direct suction of the cross flow are analysed. The position of the modifications including flow suction is defined on base of the results of the unmodified rotor blade.

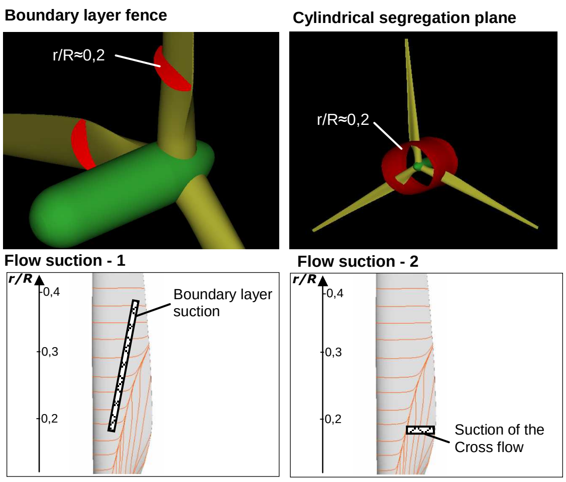

Fig. 3, Analysed modifications

4. Results and Discussion

3.1 Unmodified Rotor Blade
Results of the calculations with the unmodified rotor blade give a detailed picture of the flow field in the hub area (Fig. 4). It shows that the cross flow is mainly fed by massive separation at the cylindrical shaped hub area of the rotor blade due to centrifugal forces.

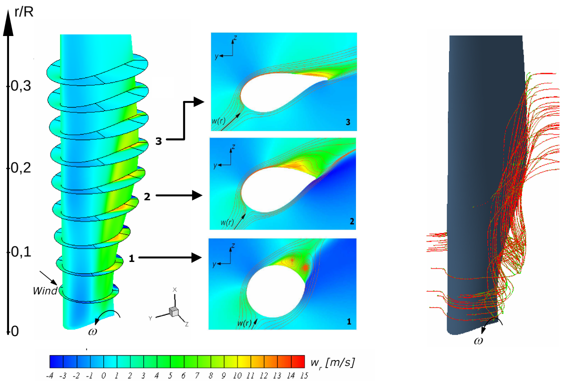

Fig. 4, Flow field in the near hub area around the unmodified rotor blade

3.2 Rotor Blade with Modifications
The effect of the described modifications on the flow at the suction side is plotted in Fig 5. On the inboard side barriers like boundary layer fences and the segregation plane show a negative influence on the flow field. Recirculation of the flow near the leading edge appears and enlarges the area of separation. On the outboard side of the smallest boundary layer fence a positive effect can be found – separation and cross flow are slightly reduced. With crowing barrier dimensions this positive effect declines and turns to a strong negative effect at the cylindrical segregation plane.
Suction of the boundary layer upstream of the separation line has a relatively strong positive effect. The maximum radial position where separation and cross flow appears is reduced significantly.
The effect of the cross flow suction is comparable to the one found by using a small boundary layer fence. In both cases cross flow and separation is reduced on the outboard side locally but the radial extension is similar to the one at the unmodified rotor blade.

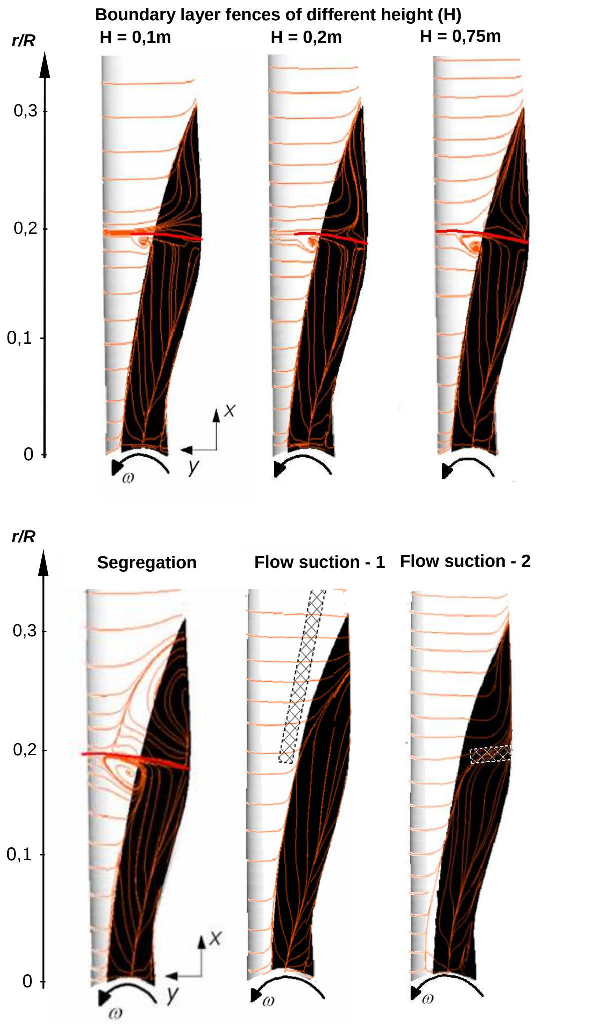

Fig. 5, Computed streamlines on the suction side of the rotor blade under influence of different aerodynamic modifications (black sections represent the area of separation and cross flow at the unmodified rotor blade)

The computed increase of the power coefficient by different modifications is plotted in Fig. 6. The results show that boundary layer fences have only a relatively small effect on the generated power of the turbine. The computed power increase of the smallest fence (H=0,1m) amounts to 0.07%. With crowing size of the fence this positive effect does not increase. The boundary layer fence with a size of H=0,75m results in a power loss. A further

enlarged barrier (cylindrical segregation plane) increases the negative effect additionally.
As already shown by the analyses of the streamlines along the surface of the rotor blade (Fig. 5) modifications including flow suction generate a positive effect on the flow field in the hub near area. They also lead to an increase of the power coefficient.
It has to be pointed out that the shown diagram shall only give a qualitative impression of the influence of investigated modifications. Due to modelling effects the exact numbers of the real power increase might be different.

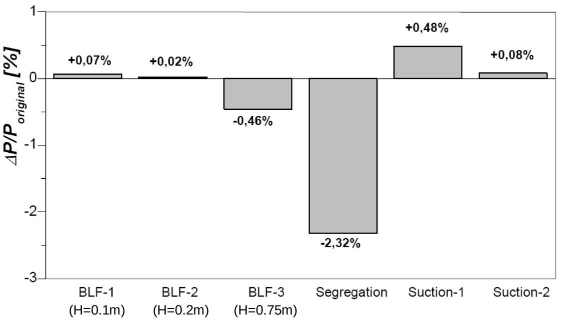

Fig. 6, Comparison of the computed power increase caused by applied modifications (BLF: boundary layer fence, Suction 1: suction of boundary layer, Suction 2: suction of cross flow)

5. Conclusion

The results of the performed numerical simulation give a deep insight in the flow field of the analysed rotor blades. Flow phenomena in the near hub section like massive flow separation and cross flow can be described in detail.
Several modifications like boundary layer fences and flow suction were implemented in the flow model. Boundary layer fences show potential as power increasing modifications. Choosing an optimum size is not a trivial task. An high number of variations have to be analysed in order to find the optimal size, position and number of those devices.
Power increase by flow suction is also possible. Influencing the boundary layer upstream of the separation line seems to be most efficient which is comparable with the installation of vortex generators.

6. References

[1] Rauch, J.; Krämer, T.; Heinzelmann, B; Twele, J; Thamsen, P.U.: 3D numerical simulation and evaluation of the air flow through wind turbine rotors with focus on the hub area. In: Euromech Colloquium 464b Wind Energy, 2005
[2] Seidel, M.: Energy output optimization of wind power stations by specifically influencing the aerodynamic behaviour of the rotor blades, Hochschule Zittau/Görlitz (FH) – University of Applied Sciences, Diplomarbeit 2004