3D CFD Quantification of the Performance of a Multi-Megawatt Wind Turbine

J Laursen, P Enevoldsen, S Hjort
Siemens Wind Power A/S, Borupvej 16, DK-7330 Brande
Email: laursen.jesper@siemens.com

Abstract. This paper presents the results of 3D CFD rotor computations of a Siemens SWT-2.3-93 variable speed wind turbine with 45m blades. In the paper CFD is applied to a rotor at stationary wind conditions without wind shear, using the commercial multi-purpose CFD-solvers ANSYS CFX 10.0 and 11.0. When comparing modelled mechanical effects with findings from other models and measurements, good agreement is obtained. Similarly the computed force distributions compare very well, whereas some discrepancies are found when comparing with an in-house BEM model. By applying the reduced axial velocity method the local angle of attack has been derived from the CFD solutions, and from this knowledge and the computed force distributions, local airfoil profile coefficients have been computed and compared to BEM airfoil coefficients. Finally, the transition model of Langtry and Menter is tested on the rotor, and the results are compared with the results from the fully turbulent setup.

1. Introduction

The aerodynamics of a horizontal axis wind turbine (HAWT) can be difficult to quantify in details due to a number of complex flow phenomena. This paper deals with the use of Computational Fluid Dynamics (CFD) to describe the performance of a SWT-2.3-93 variable speed wind turbine. During this first introduction, a number of references are referred to. The introduction should not be regarded as a complete literature review upon wind turbine modelling, but rather as a collection of references that has served as inspiration for the present work.

The rotor aerodynamics of a HAWT is influenced by rotational wake effects downstream of the rotor and effects of wind blockage upstream of the rotor. Closer to the rotor, three-dimensional effects originating from a combination of radial flow and coriolis forces play an important role on the inboard part of the blades, where the momentum in the boundary layer is weak (Figure 1, right). In general the three-dimensional effects on this part of the rotor have a positive effect on the lifting performance of the inboard blade sections as compared to 2D airfoils characteristics. This phenomenon is often referred to as stall-delay; it was first described in [1] and has been further investigated in other studies, e.g. [2-4]. Additionally, highly three-dimensional flow is occurring at the blade tips due to the formation of tip vortices (Figure 1, left). Several variable external factors such as blade surface roughness, atmospheric background turbulence and turbulence generated from nearby located wind turbines complicates the aerodynamics of a wind turbine even further. Finally, a HAWT is a dynamic structure that constantly undergoes structural deformations which in turn influence the aerodynamics.
Traditional HAWT blade design is often based on blade element momentum (BEM) theory. In BEM models the accuracy is highly dependent on the application of correct airfoil data. The airfoil data are mainly delivered in the form of lift and drag coefficients determined through wind tunnel experiments or dedicated codes like e.g. the XFOIL code [5]. In BEM computations two-dimensionality is assumed such that the flows at adjacent spanwise blade sections are unaffected by each other. This assumption is generally valid for the main part of the blade, but in the areas described above where three-dimensional flow is present the BEM model is not applicable without the introduction of tip loss corrections and additional empirical corrections accounting for the three-dimensional flow at the inboard section.

During the last decade, CFD modelling of wind turbines have evolved from scientific work performed at research institutions towards investigations performed at wind turbine manufacturers with the application of commercial codes. In several benchmark studies like e.g. the NREL phase VI rotor experiments, CFD has proven its ability to reproduce the experimental results [6-11]. Of the above-listed investigations, [7] reproduced the results best. [8] applied the more advanced DES model for a parked rotor blade, but the results did not seem to justify the choice of DES over RANS turbulence modelling.

With the growth in experience of applying CFD to wind turbines and the increment in available computing power, a number of CFD investigations have been performed on larger wind turbine blades. Often these investigations have been focused on quantifying some of the complex flow phenomena that have not been reliably quantifiable by other methods. In [12] a tip study was performed on a smaller wind turbine blade where the performance of a swept tip versus a standard tip was investigated and compared to existing tip loss models. In [13] winglets on a 50m blade were investigated, and the potential for extra power production with 5 different configurations was estimated using rotor CFD. A new blade design with increased chord and twist at the inboard sections combined with an egg-shaped nacelle was analyzed in [14]. The manufacturer of the investigated design was claiming a considerable higher yield, but the CFD investigations only showed a slight increase in Cp.

Until recently, steady state solutions have been the standard solution in HAWT rotor CFD computations, but lately the need for estimating rotor performance and especially rotor loads during operation in wind shear or in yaw has pushed modellers towards transient solutions [15]. Transient computations with a full resolution of the boundary layer on the blades are very time consuming, however, and their use as real-life design tools will probably remain limited for a while yet.
Another branch of CFD solutions that combine BEM codes with traditional Navier-Stokes solvers has been successfully applied for wake computations. Instead of modelling the blade directly, equivalent volume forces are applied on either a permeable disc or along lines at the position of the blades in the actuator disc and actuator line models. The application of actuator models has made transient computations affordable, and these models are especially suitable for estimating wake effects and the resulting loads in wind farms [16-20]. However, for the purpose of modelling different blade designs directly without prior knowledge of the proper section coefficients along the blades the full CFD solution remains preferable.

The investigations mentioned above are all examples demonstrating that CFD is already a mature tool for investigating the aerodynamic performance on large-scale wind turbines. Some outstanding issues remain, however, that demand alertness of the modeller. The main problem associated with CFD modelling of wind turbine blades originates from the well-known fact that the RANS turbulence models tend to over-predict blade performance during stall. The importance of this problem is diminishing, since most of the wind turbines existing on the commercial markets are pitch regulated where the blades, except for the innermost part near the root section, are operated well below stall.

In conclusion, nowadays CFD offers the industry an opportunity to achieve accurate information about the aerodynamic performance of pitch operated HAWT blades, including three-dimensional effects.

In the present paper a multi-megawatt HAWT rotor is modelled with the commercial ANSYS-CFX code, and mechanical power output and distributions of axial and tangential forces are extracted from the model and compared with related BEM code results. Results from parallel CFD computations performed by Risø National Laboratory, Denmark [21], are also compared. The modelled mechanical power output of the rotor is compared with measurements from a test turbine. Additionally, airfoil coefficients are extracted from the model and are compared with the airfoil characteristics of the BEM code used for everyday power performance calculations. Finally the transition model of Langtry and Menter [22-24] is tested on the blade and the outcome is compared with the fully turbulent case.

2. Method

All computations have been performed with the commercial general purpose CFD codes, ANSYS CFX 10.0 and 11.0. ANSYS CFX utilizes a finite-volume based unstructured parallelized coupled algebraic multigrid solver with a second order advection scheme and second order overall accuracy [25]. The computations have been performed with the incompressible version of the Reynolds Averaged Navier-Stokes (RANS) equations and the SST [26] turbulence models. The rotor computations have primarily been performed with a fully turbulent boundary layer, but more recent simulations with the application of the Langtry and Menter correlation based transition model [22-24] are also included. When applying the Langtry and Menter transition model, two additional transport equations for the intermittency $\gamma$ and the transition onset Reynolds number $Re_{\theta_t}$ are solved along with the standard RANS equations. Any correlation for determining the value of $Re_{\theta_t}$ can in principle be incorporated into CFX by the user, but here the standard correlations delivered with the model are used. The default correlations in the model are proprietary by ANSYS and therefore not known in detail by the user. In general the default correlation for $Re_{\theta_t}$ is based on the freestream turbulence intensity and the pressure gradient outside the boundary layer. The value of $Re_{\theta_t}$ determined outside of the boundary layer is diffused into the boundary layer by a standard diffusion term. The physics of the transition process is not directly modelled by the two additional transport equations. Instead, the physics of the transition process is entirely contained in the underlying experimental correlations. In the standard formulation the model can take both natural and bypass transition into account. As well, the model is also suited for modelling relaminarization and turbulent reattachment. Also surface roughness effects can be taken into account when using the transition model.

All computations have been run in parallel on the in-house computing cluster at Siemens Wind Power. In order to be able to run more than one computation at a time, the cluster has been split up into sub groups. On one group of 9 Pentium4 machines, the fully turbulent cases were converged in
approximately 12 hours, whereas the setup including the transition model converged within approximately 2 days of computations.

In order to save computational resources, 120 degree periodicity was applied, and only one blade was modelled. The use of periodicity prevents the possibility to include wind shear or yaw errors in the model. The tower was not modelled, since for an upwind turbine it is a fair approximation to neglect the effect of the tower on the rotor aerodynamics. All rotor computations are stationary, performed at constant uniform wind speeds and constant pitch and RPM, i.e. any unsteady features in the incoming flow and variation in turbine operation are neglected. The computations are performed both with and without a transition model so the boundary layer is modelled both with free transition and as fully turbulent, depending on the setup.

A block structured hexahedral mesh is applied for the main part of the mesh and a y+ smaller than approximately 2 is utilized for most of the blade in order to ensure a well resolved boundary layer and fulfilment of the SST turbulence model criteria (Figure 2). A C-mesh is applied, since this fits well for the outermost part of the blade. In the spanwise direction 100 grid points are located along the blade, and a resolution of 160 grid points are applied in the chordwise direction. An expansion ratio of 1.2 is applied for the directions away from the blade. At the rotational centre of the domain an unstructured mesh is used. The computational mesh extends ten rotor radii upstream and downstream of the turbine to ensure that the flow at the turbine is unaffected by the presence of the outer boundaries. Likewise, the domain extends ten rotor radii in the radial direction.

![Figure 2. Computational mesh. Complete domain (left) and zoom in on blade and nacelle (right).](image)

Due to the long computational times and hardware restrictions a detailed grid resolution dependency study has not been performed. Experience from own 2D computations, 3D computations reported by other authors and comparisons with field measurements indicate that the present grid size is sufficient for obtaining accurate results on the mean torque and force distributions.

In the circumferential direction periodic boundaries are applied. A steady inlet velocity boundary with low turbulence intensity is applied at the upstream axial boundary, and opening boundaries with atmospheric pressure are applied to the appropriate outlet boundaries. On the blade and nacelle no-slip wall boundaries with a smooth surface are applied.

3. Results

By monitoring the torque $T$ about the axis going through the centreline of the nacelle and multiplying with the angular velocity $\Omega$, the mechanical power $P$ of the wind turbine was calculated:

$$P = T\Omega$$
Table 1 shows comparisons between computed and measured mechanical power output. The measurements in table 1 values are taken from a power curve measured by Risø in accordance to IEC 61400-12 at the Høvsøre test-stand 5 close to the North Sea shore in Denmark [27]. The measured mechanical power outputs are computed by calculating back from the electrical power with the application of appropriate loss coefficients for energy loss in gear, generator and frequency transformer.

Table 1. Mechanical power output [kW]. Ellipsys is the CFD code of Risø and DTU, Denmark, and the computations are taken from a consultancy job performed by Risø for Siemens Wind Power [21]. Xblade is an in-house BEM code.

| Wind speed | RPM | Measured | ANSYS-CFX Fully turb. | ANSYS-CFX Trans. model | Ellipsys | Xblade |
|------------|-----|----------|----------------------|----------------------|----------|--------|
| 6 m/s      | 10.0| 400      | 396                  | 428                  | 392      | 408    |
| 8 m/s      | 13.5| 986      | 950                  | -                    | 945      | 967    |
| 10 m/s     | 16.0| 1894     | 1853                 | 1977                 | 1850     | 1850   |
| 11 m/s     | 16.0| 2323     | 2388                 | -                    | -        | -      |

As indicated in table 1, good agreement between measurements and modelling results are obtained with the three codes. It is also seen that the fully turbulent case slightly under-estimates the power compared to the measured values, whereas the inclusion of the transition model results in slight over-predictions. The trends are also shown by the force plots in figure 3. From figure 3 it is evident that the two CFD models with the fully turbulent boundary layer (CFX fully turb. and Ellipsys in figure 3) predict quite similar force distributions along the blade. The predictions by the BEM code show somewhat different distributions with higher force on the outer part of the blade and less on the inner part. This trend has also been reported by [28], where it was shown that the tip loss and 3D effects at the inner part of the rotor were underestimated in BEM computations compared to actuator disc computations. The results also indicate that the tip loss is underestimated by the BEM version used here. In Xblade, a combination of the 3D correction model of [29] and a reduced Prandtl tip loss has been applied on top of 2D C₁ and C₂ polars obtained from wind tunnel measurements, since this has proven to fit well with the measured wind turbine performance. Despite of the differences in modelled force distributions, Table 1 shows that the total integrated mechanical power is fairly well predicted by all the models, which indicates that the underestimation at the inboard part of the rotor is counter balanced by the overestimation at the tip. Both tip loss and 3D effects at the inner part of the blade are features that are inherently included in a CFD rotor computation (Figure 1). The inclusion of the transition model in CFX results in higher loadings all along the blade, with the most significant increase at the inboard sections (Figure 3). This issue will be further discussed below.
Figure 3. Modelled force distributions along the B45 blade at different wind speeds.

Besides delivering quantitative results like in table 1 and figure 3, CFD gives the possibility to deliver qualitative information on the rotor flow and the general mode of operation of the rotor. Examples of such information are visualizations of the induced velocities in the vicinity of the turbine or 3D streamline plots as shown in figure 4.
Figure 4. Visualization of induced axial velocities (left) and 3D streamline plot at 8m/s

Figure 4 (left) shows a contour plot of the induced velocities up- and downstream of the SW-2.3-93 rotor at 8m/s, where the blockage of the wind in front of the rotor and the complex flow in the wake is visualized. The right-hand side of figure 5 shows the matching streamline plot where flow attachment is visualized on the main part of the blade. A small pocket of separation can be seen on the inboard section.

In order to determine the three-dimensional airfoil characteristics of the blade a simple routine for obtaining the local angles of attack and profile coefficients, $C_L$ and $C_D$, has been implemented in the post processing of the results. Basically, the method is based on the reduced axial velocity method, as applied in e.g. [30]. In CFX the rotor plane is divided into annular elements in which the average axial and tangential velocities are retrieved by area integrations. From axial and tangential velocities in the rotor plane the axial and tangential induction factors $a$ and $a'$ can be determined. Figure 5 shows the computed axial induction factors for the two CFX setups and the Xblade BEM model.

$$\phi = \tan^{-1} \left( \frac{(1-a)W}{(1+a')\Omega} \right) = \tan^{-1} \left( \frac{V_y}{V_x} \right)$$

Figure 5. Modeled axial induction factors $a$ at 6m/s (left) and 10ms (right).

From knowledge of the free stream velocity $W$ and the rotational velocity in $\Omega$, the local flow angle $\phi$ can be calculated:
Where $V_y$ and $V_x$ are the local axial and tangential velocities in the annular elements in the rotor plane, as seen by the local airfoil sections. By subtracting the local twist angle $\psi$ and the global pitch angle $\theta$ from $\Phi$, the local angle of attack is found as:

$$\alpha = \phi - (\theta + \psi)$$

From the force distributions along the blade extracted from the CFD solution, the local lift and drag coefficients $C_l$ and $C_d$ are calculated as:

$$C_l = C_y \cos \phi - C_x \sin \phi$$
$$C_d = C_y \sin \phi + C_x \cos \phi$$

Where $C_x$ and $C_y$ are the force coefficients in the tangential and normal directions, respectively.

Figure 6 presents plots of $\alpha$, $C_l$ and $C_d$ for the setups with both the transition model and fully turbulent boundary layer at wind speeds of 6, and 10 m/s.

Figure 6. Modelled angle of attack, $C_L$ and $C_D$ along the B45 blade at 6 m/s (left) and 10 m/s (right).
From figure 6 it is apparent that on the inboard sections the calculated angle of attack is higher in CFX than in Xblade. However, there is no significant difference in angle of attack between the fully turbulent setup and the case where the transition model is included. The CFD model clearly predicts higher lift coefficients at the inboard sections, which is most likely a consequence of three-dimensional effects. Furthermore, inclusion of the transition model results in higher values than the fully turbulent setup. This trend corresponds well with what has also been seen from other two-dimensional investigations and wind tunnel measurements where large differences in airfoil performance, particularly for thick airfoils have been observed depending on whether transition is triggered at the leading edge or transition is kept free. The trend with higher $C_L$ when applying the transition model is apparent all along the blade. Again, this corresponds well with measurements where it is clear that a triggered boundary layer results in a decrease in slope on the lift polar curve and a corresponding lower value of $C_{L,max}$. Near the blade tip lower lift is predicted by CFD than by the BEM code, indicating that the tip loss correction of the BEM code is too optimistic. As expected, when calculating with a fully turbulent boundary layer (Figure 6 and 7) the calculated drag is higher than what is typically observed in wind tunnel experiments on smooth airfoils. Applying the transition model clearly lowers the drag on the blade, with a corresponding better agreement between the CFD model and the BEM model with 2D airfoil characteristics obtained from wind tunnel measurements.

From figure 7 it is also observed that the drag predicted by the CFD code is significantly increased near the tip, which is a direct consequence of the induced drag originating from the roll up of vortices at the tip.

The difference in computed airfoil coefficients is further elaborated in figure 8, where extracted pressure coefficients $-C_p$ and skin friction coefficients $C_f$ are shown at the positions of $r/R=0.25$, $r/R=0.50$ and $r/R=0.75$ at a wind speed of 6m/s.

**Figure 7.** Zoom in on the modelled $C_D$ along the SW-2.3-93 blade at 6m/s (left) and 10m/s (right).
For the $r/R=0.50$ and $r/R=0.75$ sections only slight differences are apparent between the fully turbulent case and the case with transition model, whereas the transition model at the $r/R=0.25$ section seems to produce a larger suction peak than when calculating fully turbulent. As expected larger differences are observed for the modelled skin friction coefficients, where the first approximately 35% of the airfoil at $r/R=0.75$ is laminar, whereas only the first 22% of the airfoil sections at $r/R=0.25$ and 0.50 are laminar. Naturally, the computations with the fully turbulent setup do not capture the jump in skin friction coefficient when the boundary layer undergoes transition from laminar to turbulent.

To give an indication on how much of the blade that is actually working under laminar conditions in normal operation, a turbulence intermittency plot is included below (Figure 9).
The intermittency plot on figure 9 shows a steep gradient in turbulent intermittency across the transition line on the blade. The boundary layer is laminar on the blue areas in figure 9, whereas red values indicate high intermittency and subsequent mixing of turbulent kinetic energy into the boundary layer. When the intermittency is introduced into the boundary layer, a production of turbulent kinetic energy will be initiated. The actual transition line will be located a little downstream of the shifting line indicated on figure 9. This is because there is a delay due to the fact that turbulent kinetic energy needs to build up before the laminar boundary layer will undergo transition, with a subsequent jump in skin friction coefficient. Note also that the stagnation point line can be observed at the innermost section on the pressure side of the blade. At the stagnation point the surface flow has increased intermittency due to flow deceleration upstream of stagnation. Downstream of stagnation the flow is accelerated and the intermittency stays at a low level.

4. Conclusions and suggestions for future work

The present investigations show the potential of CFD use in the commercial wind industry for the evaluation of blade performance.

At this stage the investigations involving the application of the Langtry-Menter transition model are at an early stage, and more work needs to be done in order to establish the most favourable setups for obtaining optimal results. In order to increase the accuracy of the simulations several of the investigations already initiated in this work should be continued.

From the computations presented above it is clear that the choice of setup with respect to applying a transition model or not has a great impact on the simulated performance of the blade. So far a fully turbulent boundary layer has routinely been assumed in wind turbine rotor modelling with CFD, and such computations have resembled measurements and expectations fairly well. The question is whether even better agreement between CFD results and actual performance of the wind turbines can be obtained. In the present analysis, introducing the transition model lowered the predicted drag to a more realistic level. At the same time the predicted lift was also increased. An increase in lift would be expected due to the known increase in slope of the lift curve around the operational point where the wind turbine is designed to be working, particularly when considering a laminar profile versus a profile with a triggered boundary layer. Whether the fairly large increase at the inboard section is realistic has to be verified by field measurements. The increase of 6-8% in mechanical power from the fully turbulent setup to the setup with transition model might also seem relatively large, but it is a direct consequence of the tendencies discussed above.

The measured mechanical power fits somewhere between the mechanical power modelled with a fully turbulent boundary layer, and the mechanical power modelled with a transition model. This indicates that the optimal setup is somewhere in between the two applied setups, but only more numerical investigations and especially field measurements are required to confirm this observation.
Finally, it should be noted that the present mesh is slightly coarser than what is recommended by ANSYS when applying the Langtry-Menter transition model. Further investigations with a finer mesh are planned in the future research at Siemens Wind Power.

References

[1] Himmelskamp H. Profile investigations on a rotating airscrew. MAP Volkenrode, Reports and Translation 1947; No. 832.
[2] Wood DH. A three-dimensional analysis of stall-delay on a horizontal-axis wind turbine. Journal of Wind Engineering and Industrial Aerodynamics 1991;37:1-14.
[3] Hu D, Hua O, Du Z. A study on stall-delay for horizontal axis wind turbine. Renewable Energy 2006; 31:821-836.
[4] Schreck SJ, Sørensen NN, Robinson MC. Aerodynamic structures and processes in rotationally augmented flow fields. Wind Energy 2007;10:159-178.
[5] Drela, M. 1989, XFOIL: An analysis and Design System for Low Reynolds Number Airfoils. Conference on Low Reynolds Number Aerodynamics. University Notre Dame, 1989.
[6] Simms D, Schreck S, Hand M, Fingersh L.J. NREL unsteady aerodynamics experiment in the NASA Ames wind tunnel: A comparison of predictions to measurements. NREL/TP-500-29494.
[7] Sørensen NN, Michelsen JA, Schreck S. Navier-Stokes predictions of the NREL Phase VI Rotor in the NASA Ames 80 ft x 120 ft Wind Tunnel. Wind Energy 2002; 5(2-3):151-169.
[8] Sørensen NN, Johansen J. Detached-eddy simulation of flow around the NREL phase VI blade. Wind Energy 2002; 5(2-3):185-197.
[9] Xu G, Sankar LN. Development of engineering aerodynamics models using a viscous flow methodology on the NREL phase VI rotor. Wind Energy 2002;5(2-3):171-183.
[10] Duque EPN, Burklund MD, Johnson W. Navier-Stokes and comprehensive analysis performance predictions of the NREL phase VI experiment. J. Solar Energy Engineering 2003; 125(4):457-467.
[11] Schmitz S, Chattot J-J. A parallelized coupled Naier-Stokes/vortex-panel solver. J. Solar Energy Engineering 2005;127(4):475-487.
[12] Hansen MOL, Johansen J. Tip studies using CFD and comparison with tip loss models. Wind Energy 2004;7:343-356.
[13] Johansen J, Sørensen NN. Aerodynamic investigation of winglets on wind turbine blades using CFD. Risø-R-1543(EN) 2006.
[14] Johansen J, Madsen HA, Sørensen NN, Bak C. Numerical investigation of a wind turbine rotor with an aerodynamically redesigned hub-region. 2006 European Wind Energy Conference and Exhibition, Athens (GR) 2006.
[15] Sørensen NN, Johansen J. Upwind, aerodynamics and aero-elasticity. Rotor aerodynamics in atmospheric shear flow. 2007 European Wind Energy Conference and Exhibition, Milano (IT) 2007.
[16] Mikkelsen R, Sørensen JN, Shen WZ. Modelling and analysis of the flow around a coned rotor. Wind Energy 2001;4:121-135.
[17] Sørensen JN, Shen WZ. Numerical modeling of wind turbine wakes. Journal of Fluids Engineering 2002;124:393-399.
[18] Mikkelsen R. Actuator disc methods applied to wind turbines. PhD dissertation, Department of Mechanical Engineering, DTU, Lyngby (DK), 2003.
[19] Troldborg N, Sørensen JN, Mikkelsen RF. Numerical simulations of wakes of wind turbines in wind farms. 2006 European Wind Energy Conference and Exhibition, Athens (GR) 2006.
[20] Sørensen JN, Mikkelsen R, Troldborg N. Simulation and modelling of turbulence in wind farms. 2007 European Wind Energy Conference and Exhibition, Milano (IT) 2007.
[21] Johansen J, Bertagnolio F, Sørensen NN. Internal report involving consultancy work between Risø National Lab and Siemens Wind Power, 2005.
[22] Langtry RB, Menter FR. Overview of Industrial Transition Modelling in CFX. *Technical Report, ANSYS* 2006.

[23] Menter FR, Langtry RB, Likki SR, Suzen YB, Huang PG, Völker S. A correlation-based transition model using local variables, part 1 – model formulation. *Proceedings of ASME Turbo 2004*, Vienna, Austria.

[24] Menter FR, Langtry RB, Likki SR, Suzen YB, Huang PG, Völker S. A correlation-based transition model using local variables, part 2 – test cases and industrial applications. *Proceedings of ASME Turbo 2004*, Vienna, Austria.

[25] Technical information regarding the ANSYS-CFX solver: http://www.ansys.com/products/cfx-advanced-solver.asp.

[26] Menter FR. Two-equation eddy-viscosity model for engineering applications. *AIAA Journal* 1994;32(8):1598–1605.

[27] Vesth A, Petersen SM. Wind turbine test: Siemens 2.3 MW Mk II power curve measurements, carried out in accordance to IEC 61400-12. *Risø-I-2398(EN)*, report, 2005

[28] Madsen HA, Mikkelsen R, Johansen J, Bak C, Øye S, Sørensen NN. Inboard rotor/blade aerodynamics and its influence on blade design. Research in aeroelasticity EFP 2005. *Risø-R-1559(EN)* 2005.

[29] Snel H, Houwink R, van Bussel GJW, Bruining A. Sectional prediction of 3D effects for stalled flow on rotating blades and comparison with measurements. *Proceedings of the European Community Wind Energy Conference*, Lübeck-Travemünde (DE), 1993.

[30] Johansen J, Sørensen NN. Aerofoil characteristics from 3D CFD rotor computations. *Wind Energy* 2004;7:283-294.