17th Australasian Fluid Mechanics Conference
Auckland, New Zealand
5-9 December 2010

Performance of a model wind turbine

P.-Å. Krogstad[1], J.A. Karlsen[2] and M.S. Adaramola[1]

Department of Energy and Process Engineering
[1]The Norwegian University of Science and Technology, Trondheim, Norway
[2]Agder Energi, Kristiansand, Norway

Abstract

The performance of a 0.9m diameter model wind turbine using the NREL s826 airfoil profile has been investigated both experimentally and numerically. The geometry was laid out using Blade Element Momentum theory (BEM). The design was tested experimentally and gave a peak power coefficient of $C_P = 0.448$ at the design tip speed ratio of $\lambda = 6$. It was found that the BEM had correctly predicted the power coefficient curve very well giving virtually identical results to the measurements, except when the turbine is operating in a deep stall mode. The thrust predicted was however consistently too low by a shift of the order of $\Delta C_T \sim 0.1$. After the model tests had been undertaken, numerical calculations were performed by means of fully 3D CFD simulations using a $k-\omega$ turbulence model. The high resolution CFD predictions (using about 3.5×10^6 grid points) reproduced the model thrust coefficient almost perfectly. The predicted power coefficients were also very close to the measurements, but somewhat overestimated at high tip speed ratios. At the design tip speed ratio the CFD over-predicted the power coefficient by merely 2%.

Introduction

Because of the considerable costs involved in developing a wind turbine, it is custumary to perform model tests to verify the performance before a full scale turbine is made. The scale of the model is necessarily quite small, being typically two orders of magnitude less than the full scale turbine. Therefore the data obtained from the model is likely to suffer from scale effects. In order to be able to trust the full scale predictions that form the basis for the design, it is therefore essential to verify that the software used also performs well in reproducing the performance at model scale.

The design phase is normally started by doing general performance studies using a Blade Element Momentum method (BEM). When a good design has been sorted out, more refined flow studies may be performed using more sophisticated software that solves the Navier-Stokes equations in a rotating frame of reference. In this way possible trouble areas may be found and the geometry modified to reduce these.

The main purpose of the study reported here was to see how well the performance of a wind turbine could be predicted at model scale using a commercially available Reynolds Averaged Navier-Stokes solver (Fluent). Therefore the actual performance of the turbine was not the main focus and the model used as reference was made as large as possible to be operated in a reasonably large size wind tunnel without too much tunnel blockage interference, the philosophy being that if the performance was affected by the walls, this effect would also be included in the Computational Fluid Dynamics method (CFD) predictions.

The philosophy is that if the CFD reproduce the model scale data quite well, this gives confidence to the scaling from model to full scale predictions provided that the numerical grid system is sufficiently refined to account for the increase in Reynolds number.

Model and experimental details

The experiments were performed in the large low-speed, closed-return wind tunnel of Dept. Energy and Process Engineering at The Norwegian University of Science and Technology. The tunnel has a test-section of 1.9 m (height) x 2.7 m (width) x 12.0 m (length). The air flow is driven by a 220-kW fan that can provide freestream velocities up to U = 30 m/s within the test section. The wind turbine model being tested was mounted on a six-component force balance. The force balance can be rotated a full 360 degrees with respect to the tunnel axis. All tests reported here were performed in a uniform flow at a turbulence level of about 0.2%.

The model wind turbine tested has a 3-bladed upwind rotor and the overall rotor diameter is 900 mm. The rotor blade geometry was designed using a blade element momentum method developed in-house. The s826 NREL wind turbine airfoil section, shown in Figure 1, (see Somers [1] for details) was used throughout the span of the blade.

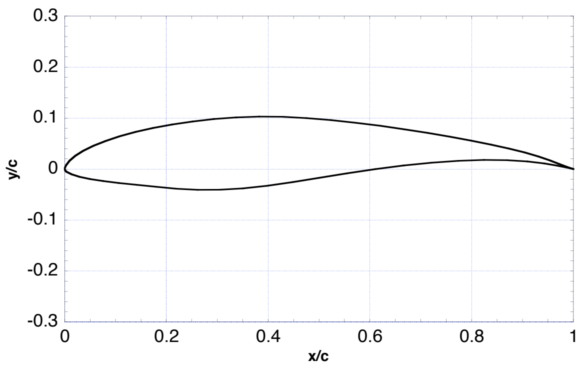


Figure 1: NREL S826 profile.

Since the main purpose for this experiment was to provide data for prediction verification, rather than to simulate the performance of a specific full-scale turbine, the blade chord length was made about 3 times wider than what is typical on commercial turbines to reduce the gap in Reynolds number. Thus, the rather solid-looking blades shown in Figure 2.

The torque generated by the wind turbine was measured by a torque sensor mounted directly on the rotor shaft. The forces on the model were obtained from the six-component force balance on which the model was mounted. Both instruments were calibrated using standard weights prior to each test. In order to compensate for the tower and nacelle thrust, the drag force acting on the tower and nacelle system was measured without the rotor blades at the same freestream velocity used for the performance measurements. This gave a parasitic drag coefficient of $C_D = 0.137$, which has been subtracted from the data presented in this report.

Figure 2: Model in the wind tunnel.

The rotor speed was controlled and monitored by a frequency inverter which was connected to a 0.37kW AC electric motor. The rotor angular speed was varied from $\Omega = 10.5$ to 249 rad./s, which gave a tip speed ratio (TSR), $\lambda = \Omega R/U_\infty$ of 0.5 to about 12 based on the freestream velocity used for the measurements (R, being the radius of the rotor).

A set of initial measurements were taken over a range of freestream velocities to check the Re dependence on the performance of the turbine. It was found that the power coefficient curves were virtually independent of velocity for $U_\infty \geq 9$m/s (see Figure 3 which shows a selection of the data obtained). The majority of the measurements were therefore performed at a nominal freestream velocity of $U_\infty = 10$ m/s. At the design tip speed ratio ($\lambda = 6$) the tip Reynolds number based on the chord length was $Re = c\Omega R/\nu = 1.0 \times 10^5$.

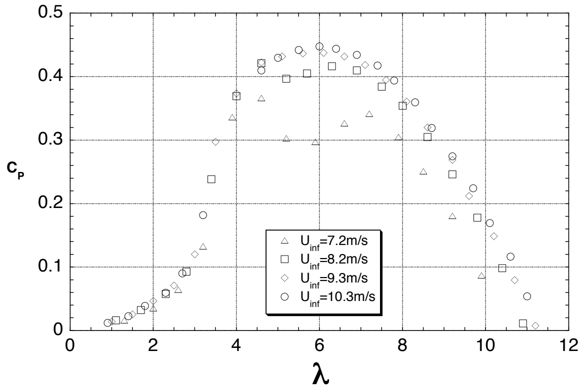
The thrust data show an almost perfect collapse for all but the lowest velocity.

Numerical simulation

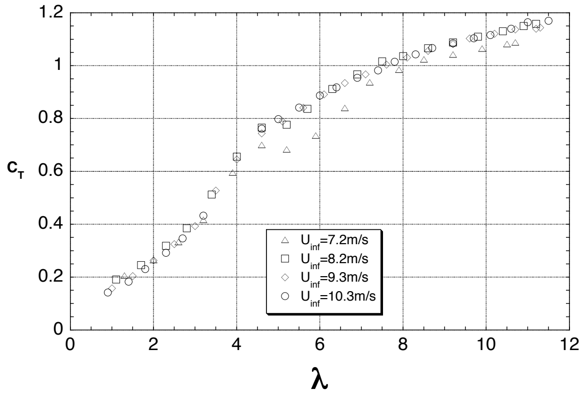
Initially a set of two-dimensional computations of the flow around the NREL s826 airfoil were performed to see how well the Fluent and XFOIL software packages would agree, and to get some information about the required grid distribution. These calculations were performed using the fully turbulent $k-\omega$ SST and the $k-\varepsilon$ RNG turbulence model options with the free stream turbulence intensity set to 1%.

After a grid independence study it was found that in the boundary layer grid the first grid point should be placed below $y^+ = 5$ from the airfoil surface. This was required because enhanced wall treatment proved to be necessary in order to get good predictions of the surface friction. It was also found that the total height of the boundary layer grid should be at least 10% of the chord length, in order to capture the flow gradients in cases when the airfoil stalled. As observed by Baxevanou and Vlachos [2], it was found that the lift coefficient was not very sensitive to the grid geometry or the number of grid cells used, while the predicted C_D was, as expected, much more sensitive.

In this way grid-independent estimates for C_D and C_L were obtained. The $k-\omega$ SST turbulence model gave results which were very close to those from the $XFOIL$ freeware, (see [3]), while the $k-\varepsilon$ RNG model with the enhanced wall function option always predicted higher C_D and C_L value than $k-\omega$ SST(Figure 4). Although C_L from $k-\varepsilon$ RNG was closer to



(a)



(b)

Figure 3: Experimental power and thrust coefficients. Reynolds number dependency tests.

the XFOIL predictions in the normal operating range, its overestimation of C_D by almost a factor 2 at the operating angle of attack ($\alpha = 7^o$) and the large difference in stall prediction led to the decision to use the $k-\omega$ SST model for the 3D simulations.

These 2D calculations gave confidence to the data that had been used in the BEM to design the rotor, but also to the choice of the three-dimensional grid system to be used in the fully three-dimensional Fluent calculations.

The 3D simulations for the model turbine were performed as steady state calculations in a rotating frame of reference, taking advantage of the symmetry planes to reduce the computational domain to one third of the rotor. The geometry of the tower was therefore not included in the calculations. The computational domain extended 4.5D upstream and 7.8D downstream of the rotor plane, which corresponds closely to the streamwise dimensions of the wind tunnel test section. Because the computations were performed in cylindrical coordinates the exact boundary conditions of the wind tunnel walls could not be used. Instead, the radius of the control volume was adjusted so that the flow area was the same as in the wind tunnel test section. The boundary condition at the outer surface was then specified as a wall. In this way it was hoped that some of the effects of the finite flow domain that could influence the measurements from the model might also be included in the computations.

A structured grid was used for the pressure and suction sides of the blade surface, whereas the rest of the turbine was meshed using triangular cell faces. About 60.000 faces were used to describe one blade and one third of the nacelle. A total of 3.5×10^6 cells were used to mesh the computational volume. The SIMPLEC algorithm was used to couple the pressure and velocity fields, and the $k-\omega$ SST model was used to compute the tur-

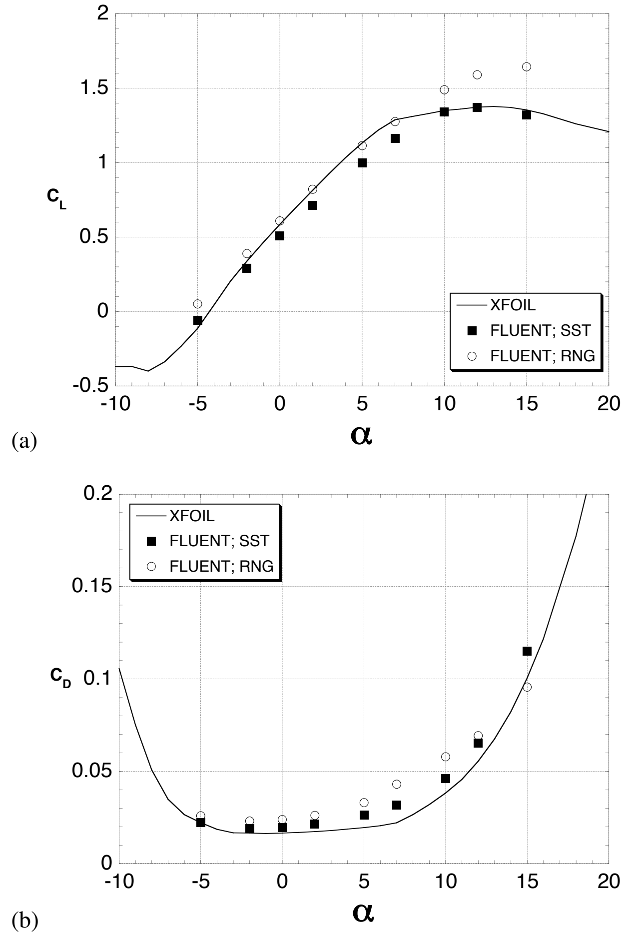


Figure 4: C_L and C_D calculated using the *XFOIL* and *Fluent* software.

bulent eddy viscosity. The momentum equations were solved using a quadratic upwind discretization scheme (QUICK) for the convective terms, while the pressure solution was discretised using a first order upwind scheme.

The simulations were performed on a 64 bit PC in parallel processor mode using 4 CPUs and took about 24 hours per case.

Results and discussion

The design calculations of the performance for the model turbine had been undertaken using the BEM method. The computations were done for tip speed ratios from λ = 2 up to where the power coefficient would first become negative. This showed that the operational range of the NREL S826 profile used would be from α = -5 to almost 35^o angle of attack. The first indications of stall were predicted to occur near the root at λ = 4, and the blade was found to be fully stalled at λ = 3. Two-dimensional predictions using the XFOIL software [3] showed that at the design *Re* the profile will have zero lift coefficient at $\alpha \approx -4^o$. This point was exceeded when $\lambda > 9$ where the inner part of the turbine then starts to feed energy into the flow while the outer section is still extracting energy.

Because of the small scale of the turbine and the complicated shape of the blade, it is quite difficult to set the blades with a precision better than about $\pm 1^o$ with respect to the rotor plane. To study the effects of blade pitch misalignment, calculations were therefore also performed for blade settings of $\pm 1^o$ with respect to the design angle of attack. The results are shown in Figure 5 and show that possible misalignment errors will affect the thrust coefficient data most severely. The power coefficient is most sensitive to misalignment near the best point of operation, while the thrust coefficient appears to be very sensitive at high tip speed ratios. It is therefore found essential to align the blades very accurately. Therefore a special alignment rig was made for the blades. Using this tool it is believed that the blades were aligned with an uncertainty of less than 0.25^o.

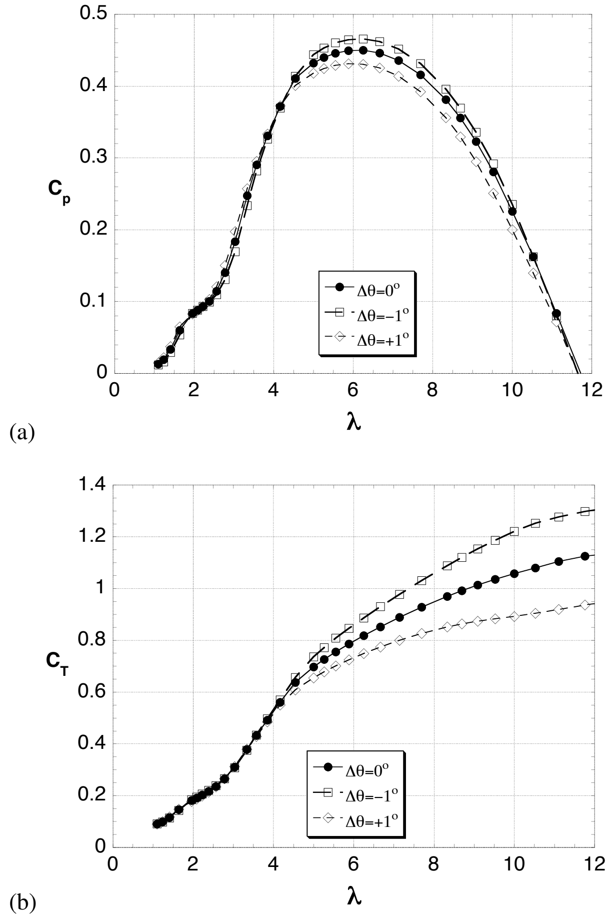
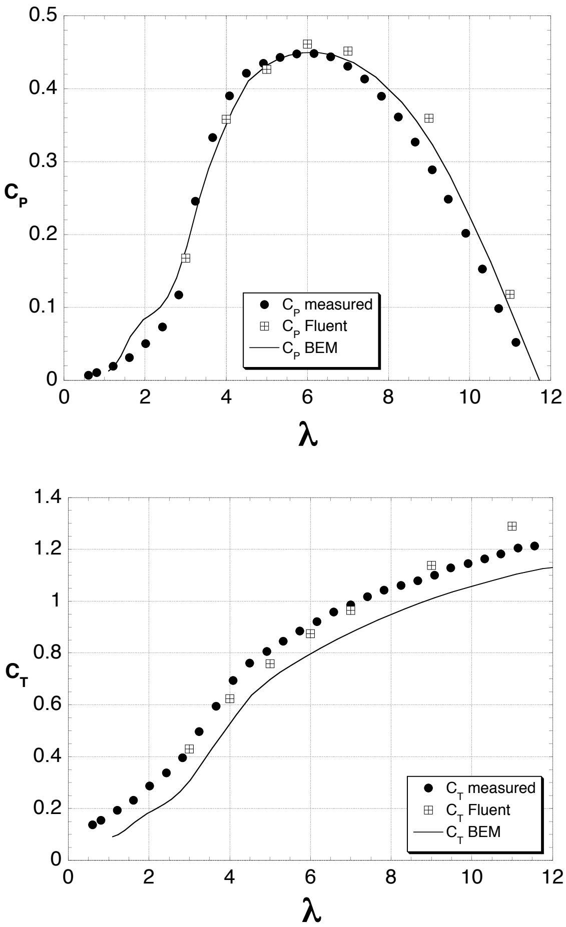


Figure 5: Performance predictions for three pitch angle settings using the BEM method.

Figure 6 shows the measured power and thrust coefficients as function of the tip speed ratio. Included in the graphs are also the design predictions from the BEM method and the CFD calculations. For the BEM method the predictions were made for a fixed *Re* of $Re = 10^5$, assuming natural transition. This *Re* corresponds to the operational *Re* at the tip for the design tip speed ratio and a test velocity of U_∞ = 10m/s. This is perhaps not a fair comparison, due to the way in which the measurements were performed. The turbine was operated at constant wind speed and this procedure was also applied for the CFD predictions. To allow λ to vary, the speed of rotation of the rotor was varied. Therefore the *Re* in the tests depends almost linearly on λ. The tip *Re* was thus changing by a factor of about 5, from about 3×10^4 for $\lambda \sim 2$ to 1.8×10^5 at $\lambda = 10$. Hence, any *Re* dependence in C_L and C_D that would be predicted by XFOIL has not been included in the predictions. However, it is assumed that this effect is not very significant and this is confirmed by the data presented.

The power coefficients predicted by the BEM method has the correct shape and follows the measured data very well except at very low λ, where the C_P estimate is too high. The estimated



(a)

(b)

Figure 6: Comparisons of power and thrust coefficients from CFD and BEM with the measurements.

peak efficiency is quite sensitive to the estimated lift and drag coefficients at the operating angle of attack. It therefore depends heavily on the assumption about the way the boundary layer develops, e.g. whether the flow is turbulent from the onset or develops more naturally. This affects primarily the drag coefficient and to a limited degree the lift in the normal operating range for the turbine. Hence, if the assumption of natural transition was not valid e.g. due to surface roughness or ambient turbulence effects, one would expect this effect to show up primarily in C_T also at the design λ, where the flow is expected to be attached across the entire blade. The C_T predicted by the BEM method is consistently too low over the entire tip speed range tested (Figure 6b), suggesting that the boundary layer predictions from XFOIL are too optimistic. Therefore the C_D predictions from the $k-\omega$ SST model in Figure 4(b) are likely to be the most reliable. But the assumption about the boundary layer development also affects the stall characteristics of the blade at low λ, where premature separation would be predicted if the assumption of natural transition was incorrect. Figure 6 suggests that the opposite is the case, since the BEM computations seem to over-estimate C_P at very low λ and therefore predicts a less severe stall behavior than is actually experienced. This might be a consequence of ignoring three-dimensional effects in the BEM method.

The CFD predictions from the Fluent simulations follow the measurements very well. The peak efficiency is over-predicted by about two percent, but the general shape is correctly captured including the onset of separation in the region for $\lambda < 4$, which is very well reproduced.

As pointed out previously, the thrust coefficient predicted by the BEM method for zero pitch angle (Figure 6b) is consistently too low, but the agreement between the CFD calculations and the measurement is excellent. As seen by comparing Figure 5(b) and Figure 6(b) it is possible that the blades on the model may be slightly misaligned compared to the design conditions. Decreasing the pitch angle by less than 1^o would be sufficient to bring the C_T predictions from the BEM method in agreement with the measurements at high λ, but this can not explain the differences at low λ. No measurable misalignment of the blades could be found.

Conclusions

The investigation presented shows that the standard blade element momentum method is capable of giving a very good impression of the performance of a wind turbine. In addition to the approximations built into the method, the performance of the method obviously relies critically on how well the performance of the airfoil sections used are known. In the present investigation the performance of the model turbine was calculated assuming natural transition. Compared to the data obtained from model wind tunnel tests it was found that the general shape of the power and thrust coefficient curves were correctly predicted, but the thrust coefficient was consistently lower than the measurements throughout the operating tip speed range. Since the thrust coefficient was under-predicted over the entire tip speed ratio, this suggests that XFOIL under-predicted the drag. This was verified by the 2D CFD simulations performed using the Fluent software. For a very slow running turbine the power coefficient was over-predicted, probably because XFOIL failed to predict the airfoil characteristics at very high angles of attack correctly ($\alpha > 20^o$).

The turbine performance was also estimated using high resolution, fully 3D CFD simulations by means of the Fluent software with the $k-\omega$ SST turbulence model. About 3.5×10^6 grid points were found necessary to give a good representation of 1/3 of the flow domain, taking advantage of the geometrical symmetry. These calculations gave an almost perfect prediction of the performance of the model turbine throughout the operational range.

Based on the information obtained from this investigation we have demonstrated that the Fluent CFD software package is able to predict the wind turbine performance at model scale quite accurately. This is important information since good agreement at model scale will give confidence to the calculations when the turbine performance is to be estimated at full scale conditions. Although the blade element momentum method predicted the behavior of the turbine very well, it has, as expected, a much higher degree of uncertainty, since it relies heavily on how well the airfoil characteristics are known.

References

[1] Somers, D.M., The S825 and S826 Airfoils. *National Renewable Energy Laboratory* 2005; NREL/SR-500-36344.

[2] Baxevanou, C.A. and Vlachos, N.S., A comparative study of numerical schemes and turbulence models for wind turbine aerodynamics modelling. *Wind Engineering*, **28**, 8, pp. 275-290, 2004.

[3] Drela, M., Xfoil v. 6.97. http://web.mit.edu/drela/Public/web/xfoil/