Numerical simulations of the NREL S826 airfoil

KF Sagmo, J Bartl and L Sætran
Norwegian University of Science and Technology, NO-7491 Trondheim, Norway
E-mail: krissag@stud.ntnu.no

Abstract. 2D and 3D steady state simulations were done using the commercial CFD package Star-CCM+ with three different RANS turbulence models. Lift and drag coefficients were simulated at different angles of attack for the NREL S826 airfoil at a Reynolds number of 100 000, and compared to experimental data obtained at NTNU and at DTU. The Spalart-Allmaras and the Realizable k-epsilon turbulence models reproduced experimental results for lift well in the 2D simulations. The 3D simulations with the Realizable two-layer k-epsilon model predicted essentially the same lift coefficients as the 2D Spalart-Allmaras simulations. A comparison between 2D and 3D simulations with the Realizable k-epsilon model showed a significantly lower prediction in drag by the 2D simulations. From the conducted 3D simulations surface pressure predictions along the wing span were presented, along with volumetric renderings of vorticity. Both showed a high degree of span wise flow variation when going into the stall region, and predicted a flow field resembling that of stall cells for angles of attack above peak lift.

1. Introduction

Numerical simulations of fluid flow problems promises a relatively cheap and fast way of assessing key characteristics of wing designs, compared to wind tunnel tests. However, they are of little value without experimental reference data. A recent NTNU study by Aksnes [1] performed measurements on the NREL S826 airfoil developed by the National Renewable Energy Laboratory in America [2], providing a size-able set of reference data for numerical simulations. The NREL S826 make up the blade profile of a model wind turbine design used in a series of ”blind test” case studies hosted at NTNU [3], and hence performance data on the wing profile may be a valuable asset for the participating community. In addition, performance characteristics of the airfoil have been measured and simulated both at the Technical University of Denmark [4, 5] and the Middle East Technical University’s Centre for Wind Energy [6, 7], allowing for a further comparison of the experimental and numerical data acquired.

The project work at hand makes use of the Computational Fluid Dynamics (CFD) software package STAR-CCM+ developed by CD-Adapco, and assesses some CFD turbulence’s models ability to accurately predict performance characteristics of the NREL S826 airfoil. Both 2D and 3D steady state simulations are used. Although it can be considered low for the application of the airfoil profile, the flow conditions are set to match a Reynolds number of 100 000, with low inflow turbulence due to the amount of experimental data available for these conditions. Previous work by Sarlak [8] and Cakmakcioglu [6], among others, included higher Reynolds number flows for a range of numerical models.

A mesh refinement study was done for the Realizable k-epsilon 2D simulations. Simulations using this model, in addition to the Spalart-Allmaras and Menter SST K-omega turbulence models followed. Further, 3D simulations was checked, to investigate effects not present in
2D simulations. A recent investigation conducted by Manolesos [9] suggested that turbulence occurring in the wake of an airfoil post stall is highly anisotropic, rendering the Boussinesq approximation assumed by the eddy viscosity turbulence models mentioned here, in their linear forms, invalid. However, Manolesos later went on to show that 3D simulations using the unsteady formulation of the Menter SST k-omega turbulence model can predict the onset of stall cells [10].

2. Methodology

2.1. Computational domains and boundary conditions

Aksnes reports most of his results without a correction for the 8 % wind tunnel blockage, so in order to be able to directly compare the results the computational domain was set up as close as possible to the actual experimental setup. However, for all but the deep stall angles of attack, these blockage effects amounts to only a few percent error(< 3%), in lift and drag measurements, and allows for comparison with the above stated references.

The outer dimensions of the 2D domain block is 2.71 m times 11.15 m. The 3D domain adds a 1.8 m depth to the 2D domain, and adds an additional inner refinement zone stretching about 5 chord lengths downstream of the wing’s trailing edge. Thus, the dimensions of the computational domain resembles the exact dimensions of the wind tunnel environment in the NTNU experiment [1]. The computational domains used are illustrated in figure 1. All domains were built using structured, trimmed hexagonal grids with prism (boundary) layer cells.

The final meshes selected for the 2D and 3D simulations is the result of a gradual mesh refinement study of 2D domains. In addition to complete domain refinement, the coherent length of the wake refinement zones was varied, to verify that the solution was fairly independent. This resulted in a wake refinement stretching more than ten chord lengths downstream of the wing’s trailing edge. Further, the pressure outlet was positioned 2 and 4 metres further downstream of the base configuration, to verify that the solution was relatively independent. Overall, in designing the grids, the guide by Spalart [11] has been helpful.

In Star-CCM+, one may specify turbulence model parameters, by specifying a turbulence intensity (TI), and turbulence length scale (TL). For the present study TI was set to 0.71% as measured in the experiments at NTNU. TL was estimated to 0.1355 m, which is 5% of the largest wind tunnel inlet parameter of 2.71 m. A similar approach was suggested by Versteeg [12] for approximation of the TL, when it is not know a priori.

The modelled medium was set to air in Star-CCM, which automatically sets properties such as dynamic viscosity, and molecular weight. All simulations were done with a velocity inlet, used to specify the Reynolds number of a 100 000, and a pressure outlet at 1 atm. The wing surface was modelled as hydraulically smooth, as justified by the measurements done by Aksnes. A smooth, non slip, wall condition was also specified at the outer walls of the computational blocks, although this is a simplifying assumption. Measurements previously conducted at NTNU’s department of Energy and Process Engineering indicated a boundary layer thickness of up to 200 mm at the positioning of the NREL S826 test-piece during experiments. This exceeds the boundary layer thickness modelled at the walls of the computational domain in the simulations. Because of this boundary layer, Aksnes used dummy sections of 300 mm that spanned the layers from the walls, and only measured forces on a 1175 mm long center section of the wing. This resulted in a small wing surface discontinuity in the experiments, in the order of 0.01 chord lengths at most, in order to physically separate the dummy sections from the measuring section. This approach was also adopted in the 3D simulations, as illustrated in figure 1, to investigate the developing flow.

The simulations were run using a segregated solver with a second order upwind convection scheme. Other modelling assumptions were isothermal flow and compressible, ideal gas, although incompressible modelling might as well be used. Indeed, an investigation of the density across
2.2. Wall treatment

All the present simulations, excluding the coarse mesh taking part in the mesh dependency study presented in table 1, were shown to have wall $y^+$ values well below 1, both for 2D and 3D. This induced a wall treatment similar to a low $y^+$ treatment, as is recommended for low Reynolds number flows [13].

The low wall $y^+$ treatment makes no explicit modeling assumptions, and sets the velocity distribution in the viscous sublayer as $u^+_{laminar} = y^+$. The velocity distribution in the logarithmic layer is set to

$$u^+_{turbulent} = \frac{1}{\kappa} \ln(Ey^+),$$

where the von Karman constant $\kappa = 0.42$, and the constant $E = 0.9$, by default. The definition of the dimensionless $u^+$ and $y^+$ stems from the usual law of the wall [12].

2.3. Wing profile

The computer modelled NREL S826 wing geometry used for the simulations are the data points received from Aknes’ measurements of the actual built wing, with a chord length of 0.45 m. It blends a 2 mm radius into the trailing edge of the original design by NREL, rather than having a tip thickness approaching zero. See Aknes [1] for a comparison of the original NREL S826 profile and the version both built and modelled.
2.4. Mesh dependency and uncertainties

For CFD work, Roache [14] presents, as an estimate for the discretization error $E_{f_1}$ of a target quantity $f_1$ in the finest of two grid levels, the relation:

$$E_{f_1} = \left\{ \frac{f_2 - f_1}{1 - rp} \right\}. \quad (2)$$

Here $f_2$ represents the target variable in the coarser mesh, $p$ represents the order of the numerical scheme and $r$ represents the refinement ratio between the two meshes. If the fine mesh has a base cell size of 0.2 units, and the coarse mesh has a base cell size of 0.4 units, the refinement ratio is 2 ($r = h_2/h_1 > 1$, where $h$ is some cell base size).

As an alternative to using the order of the numerical scheme, Roache also suggested using the observed order $p_{obs}$ of the truncation rate decay. Starting from a coarse mesh number 3, and gradually refining with a constant refinement ratio to mesh number 1, we have

$$p_{obs} = \ln\left( \frac{|f_3 - f_2|}{|f_2 - f_1|} \right) / \ln(r). \quad (3)$$

Further, to put some upper bounds on the discretization error, Roache introduced the grid convergence indicator GCI, such that $(GCI)U = F_S E_U$, for some target variable $U$, with a discretization error $E_U$. In this relation, $F_S$ is a safety factor. Roache’s recommendation was a safety factor of 3 for conservativeness. A less conservative safety factor of 1.25 was suggested for work where the discretization error is evaluated using $p_{obs}$.

Calculating $p_{obs}$ using eq. 3, for both the lift and drag coefficients given in table 2, and taking the mean, we arrive at $p_{obs} = 2.2$, rounded off to one decimal. Setting a suggested safety factor of 1.25 we arrive, via eq. 2 at a $GCI_{Cd}$ of about 1.3 % and a $GCI_{Cl}$ of about 0.3% for the drag and lift coefficients, respectively.

In the results later presented uncertainty due to iterative convergence, after a flat-lining of residuals was found to be negligible. Simulations usually converged within 2000 iterative steps, and readings at 8000 steps usually found a difference less than 0.2 %. This was however not true for the 3D simulations in the stall region, where values could differ with around 1.3 % for the last 100 iterations.

Table 1: 2D and 3D grid details. Grid n.3, n.2 and n.1 denote the coarse, medium and fine 2D meshes, respectively. Presented from left to right then, is the grid type, drag and lift coefficients, $y^+$ maximum value, the wing surface span wise cell resolution in terms of the chord length, the wing surface chord wise cell resolution in terms of chord length, the innermost wake refinement zone cell resolution in terms of chord length, the total element count in the specific grid (mainly cell count) and the standard CPU solver time per iteration.

| Grid | $C_d$ | $C_l$ | $y_{max}^+$ | $\delta_{span}$ | $\delta_{chord}$ | $\delta_{w.cell}$ | $\#_{tot}$ | $\Delta_{CPU}$ |
|------|------|------|------------|----------------|-----------------|-----------------|-----------|-------------|
| n.3  | 0.0675 | 1.4276 | 3.760 | - | (0.736-0.093)% | 2.772% | 5.88 · 10^4 | 0.56 s |
| n.2  | 0.0651 | 1.4411 | 0.227 | - | (0.421-0.052)% | 1.386% | 1.89 · 10^5 | 2.08 s |
| n.1  | 0.0646 | 1.4450 | 0.078 | - | (0.239-0.003)% | 0.692% | 6.81 · 10^5 | 9.44 s |
| 3D   | 0.0865 | 1.4802 | 0.238 | (0.74-0.24)% | (0.75-0.02)% | 1.390% | 3.58 · 10^8 | 457.95 s |
2.5. Computing system specifications
Most of the 2D simulations were run on a desktop computer (Windows OS) with 16 GB RAM and utilizing 5 of 6 cores on a CPU at 3.5 GHz each. 3D simulations were run on NTNU’s high performance cluster ”Vilje”, reserving 5 computing nodes containing 16 CPU cores running at 2.6 GHz each. The 3D grid generated about 35 million cells, and it took about 2 hours and 30 minutes for the simulations to converge.

See table 1 for a summary of grid parameters and run times for the Realizable k-epsilon turbulence model.

3. Results and Discussion
3.1. Results for lift and drag coefficients
The following figures, 2 and 3, illustrates the results of the final simulations of the lift and drag coefficients for a Reynolds number of 100 000, respectively. DTU experimental data by Sarlak [5] was obtained using a wall to wall wind tunnel setup, as in the NTNU experiments by Aksnes [1], and is reproduced here to illustrate the spread in the experimental values available for the wing. Because the simulation results here are presented alongside experimental values obtained from direct force measurements and also surface pressure distributions, forces from both pressure and shear are included in the CFD results. In figure 4 pressure distributions from the 3D simulations are compared with the NTNU measurements. It is here observed that the 3D Realizable k-epsilon simulations and the experimental results are in fair agreement, both pre- and post-stall.

![Lift Coefficients for the NREL s826 at Re 100 000](image)

Figure 2: Lift coefficients for the wing plotted against AoA.

From the graph presented in figure 2 it may seem that the lift coefficient curve obtained at
NTNU is slightly shifted by about 1.5 degrees, with respect to the DTU experiments. However, the large eddy simulations (LES) conducted by Sarlak [5] predicts lift coefficients that align better with the lift coefficients measured at NTNU, than DTU, and lift peaks at around 13.5 degrees. Also, the measured uncertainty in the angle of attack in the NTNU experiments were given to ±0.25 degrees, so a 1.5 degree systematic shift seems unlikely.

![Drag Coefficients for the NREL s826 at Re 100 000](image)

**Figure 3:** Drag coefficients for the wing plotted against AoA.

In general, the results show good lift predictions by the simulations, as long as the flow stays fairly attached. Compared to the NTNU experiments, lift is slightly under predicted. Compared to the DTU experiments lift coefficients are well predicted, before stall. The onset of stall is observed at a significantly smaller angle of attack in the DTU experiment, compared to the NTNU experiment and simulations. Due to the increasing 3D properties of the flow for higher angles of attack, the 2D simulations were only conducted up until an angle of 14.5 degrees. The 2D k-omega SST simulations would not converge beyond an angle of 13.5 degrees, due to the inherent instability predicted. The 3D Realizable k-epsilon simulations were conducted for even higher angles of attack, and give reasonable predictions of lift when compared to the NTNU force gauge experiments, well into the stall region, as well as for mid-span chord-wise pressure distributions as is presented in figure 4. This invites a qualitative investigation of the predicted flow structure, presented in the next section.

The NTNU data was obtained with a 60 second time sampling for each angle of attack, while the DTU experiments sampled data within a 10 second interval for each angle of attack. The difference in measured lift beyond stall may, in part, be down to the difference in sampling
time between the NTNU and DTU experiments. Also, the wing used in the NTNU experiment had an aspect ratio of 4, while the wing used in the DTU experiments had aspect ratio of 5. Considering previous findings regarding stall cells by both Winkelmann and Barlow [15] as well as Yon and Katz [16], a wing with an aspect ratio of 4 could develop both a single and double stall cells, while a wing with an aspect ratio of 5 would be more inclined to developing a pair of stall cells.

Figure 4: Pressure coefficient distribution for three different angles of attack, comparing measured values with the results from the 3D Realizable k-epsilon turbulence model simulations.

In figure 3 we note a fair overlap for the $C_d$ predictions by the 2D Spalart-Allmaras and the 3D Realizable k-epsilon turbulence models with respect to the experimental values obtained by surface pressure integration at NTNU, which would mean an under prediction of total drag by the simulations. However, the drag coefficients measured at DTU, obtained from measuring the velocity deficit in the wake, are in fair agreement with both the 2D Spalart-Allmaras and the 3D Realizable k-epsilon simulations for low angles of attack. The remaining 2D simulations seem to give slightly low estimates for drag between -5 and 5 degrees angle of attack. The high pressure drag estimates of the NTNU experiment, with respect to simulations and the DTU experiment, is in agreement with the higher prediction of lift. When entering the stall region, all simulations are observed to under predict drag.

3.2. 3D simulations - Surface pressure and flow visualizations

In figure 5 we see the span wise variation of surface pressure predicted in the simulations. Note that for AoA=8.5, very little span wise variation is predicted. For the AoA=14.5 deg we see the resemblance of a single stall cell formation. This is in agreement with the tuft visualization experiments performed at NTNU [1]. The dummy sections of the wing on the top and bottom, not part of the force measurements of the lift and drag coefficients for the wing, absorb most of the diverse flow field generated by the wall/wing interaction. This is illustrated in the flow vorticity visualization presented in figure 6. Considering the wing’s low aspect ratio of roughly 4, the numerical prediction of a single stall cell formation for a wall to wall wing is in agreement with previous findings [15, 10]. Studies by Yon and Katz [16] suggested that wing surface pressure may not be so dependent on the unsteady von Karman vortex street in the far wake region of a wing, which suggests that a steady state simulation predicting a stall cell like formation is not entirely strange. Indeed, a study by Rodriguez and Theofilis [17] summed up a lot of the previous work on stall cells, and presented a quantitative description of a wing’s flow field topology as a perturbed steady 2D flow, resulting in periodic span wise variations in a wings separation.
Figure 5: Wing surface pressure visualizations using iso curves and colored according to pressure coefficient $C_p$ for angles of attack; A: 8.5 deg, B: 14.5 deg, C: 19.5 deg, D: 14.5 deg. Red color indicates a $C_p$ near 1, blue a value near -7. Leading edges are facing inwards toward letters A, B, C, and D, respectively. Letters S and P indicates suction and pressure sides, respectively. D is the same simulation as for B, but with a continuous wall to wall wing geometry.

Figure 6 points out two separate vorticity shear layers, as presented in the time averaged flow field model for a stalled wing, by Winkelmann and Barlow [15]. The relatively short stream wise extent of the vortexes generated are in agreement with the known issues for RANS based CFD simulations regarding high vorticity diffusion and correction models for this has previously been suggested [18].

Figure 6: Volumetric rendering of the predicted flow vorticity with a magnitude between 20 and 100 s$^{-1}$. A: AoA=14.5 deg. B: AoA=19.5 deg. The volume sample is from the wall down to mid-span, to show the cross section profile of the flow predicted.
It can also be noted a thin, stream-wise vortex originating at the leading edge of the wing, where the measuring section is separated from the wall dummy section. Another set of simulations were run, with a continuous wall to wall wing, to check that this vortex did not significantly alter the stalled flow field. The surface pressure distribution remained essentially the same for both 14.5 and 19.5 angles of attack, as for the discontinuous wing. A comparison for 14.5 degrees angle of attack is given in figure 5. The main features of the vorticity field also remained the same, though the leading edge stream-wise vortex was removed. A comparison between the predicted lift, at an angle of attack of 14.5 for the two cases, revealed only a 0.6 % difference. However, a 12 % difference in drag was simulated. Here, the continuous wing generated the the most amount of lift, and least amount of drag, according to the simulations.

4. Conclusion
Considering the spread in the available experimental values [1, 4, 6] all turbulence models can be said to give fair predictions for lift and drag coefficients, though with a tendency to under predict drag, for a Reynolds number of 100 000. The good agreement for the lift coefficient predictions between 2D simulations, 3D simulations, direct force measurements and mid span surface pressure integrations are supported by the low span wise flow variations predicted for small angles of attack. The significant variations in lift from direct force measurements and mid span pressure integration in the stall region, for angles of attack above 9 degrees, can be related to the existence of inherent 3D flow effects on the wing. This study shows that while stall cells may be inherently unstable effects, the steady state solution predicted by the Realizable k-epsilon turbulence model in early stall conditions qualitatively agrees with the flow field model suggested by Winkelmann and Barlow [15], as well as the more recent findings by Manolesos [10].

References
[1] Aksnes N Y 2015 Performance Characteristics of the NREL S826 Airfoil Master’s thesis NTNU Trondheim, Norway
[2] Somers D M 2005 The s825 and s826 airfoils Tech. Rep. SR-500-36344 NREL
[3] Sætran L and Bartl J 2015 Invitation to the 2015 ”blind test 4″ workshop
[4] Sarmast S and Mikkelsen R 2013 The experimental results of the nrel s826 airfoil at low Reynolds numbers
  Tech. Rep. urn:nbn:se:kth:diva-120683 KTH
[5] Sarlak H, Mikkelsen R, Sarmast S and Sørensen J N 2014 J. Phys.: Conf. Series 524
[6] Cakmakcioglu S C, Sert I O, Tugluk O and Sezer-Uzol N 2014 J. Phys.: Conf. Series 012028 524
[7] Ostavan Y, Amiri H and Uzol O 2013 Conference on Wind Energy Science and Technology-RUZGEM
[8] Chivee H S, Sørensen J N, Mikkelsen R F and Shen W Z 2014 Large Eddy Simulation of Turbulent Flows in Wind Energy Ph.D. thesis DTU Wind Energy
[9] Manolesos M and Voutsinas S 2014 Phys. of Fluids 045101 26
[10] Manolesos M, Papadakis G and Voutsinas S 2014 J. Phys.: Conf. Series 012068 555
[11] Spalart P R 2001 Young-person’s guide to detached-eddy simulation grids Tech. Rep. CR-2001-211032 NASA, Langley Research Center
[12] Versteeg H K and Malalasekera W 2007 An Introduction to Computational Fluid Dynamics 2nd ed (Harlow: Pearson Education)
[13] CD-adapco StarCCM+ user guide code version 10.02.012
[14] Roache P 1997 Annu. Rev. Fluid. Mech. 29 123–60
[15] Winkelmann A E and Barlow J B 1980 AIAA Journal 18 1006–1008
[16] Yon S A and Katz J 1998 AIAA Journal 36 305–312
[17] Rodriguez D and Theofilis V 2010 Theor. Comput. Fluid Dyn. 25 105–107
[18] Löhner R 2009 AIAA paper 2009-135