CFD aerodynamic analysis of non-conventional airfoil sections for very large rotor blades

G Papadakis¹, S Voutsinas¹, G Sieros², T Chaviaropoulos²

¹ National Technical University of Athens, Laboratory of Aerodynamics, Greece
² Center for Renewable Energy Sources & Saving, Greece

spyros@fluid.mech.ntua.gr

Abstract. The aerodynamic performance of flat-back and elliptically shaped airfoils is analyzed on the basis of CFD simulations. Incompressible and low-Mach preconditioned compressible unsteady simulations have been carried out using the k-w SST and the Spalart Allmaras turbulence models. Time averaged lift and drag coefficients are compared to wind tunnel data for the FB 3500-1750 flat back airfoil while amplitudes and frequencies are also recorded. Prior to separation averaged lift is well predicted while drag is overestimated keeping however the trend in the tests. The CFD models considered, predict separation with a 5° delay which is reflected on the load results. Similar results are provided for a modified NACA0035 with a rounded (elliptically shaped) trailing edge. Finally as regards the dynamic characteristics in the load signals, there is fair agreement in terms of Str number but significant differences in terms of lift and drag amplitudes.

Key words: Flat-back airfoils, blunt trailing edge, elliptical airfoils

1 Introduction
The interest in investigating the aerodynamic performance of non-conventional airfoil shapes is driven by the design requirements of very large rotor blades for wind turbines in the order of 10-20MW. Rotors of this size are expected to have reduced solidity for weight optimization purposes. The reduction of the plan-form chord calls for profiles with higher design lift and higher thickness at the root region in order to undertake design loads. Reduction of rotor performance (and therefore loading per unit area) in the benefit of a larger diameter and swept area is an alternative option which is presently under consideration for the very large offshore turbines.

The present work addresses the first option by analysing the aerodynamic performance of non-conventional airfoil shapes. Assuming that reduction of solidity mainly affects the chord, thicker airfoils are produced resembling the sections over the inboard part of the blades. In this region the sections have blunt trailing edge (also known as flat-back airfoils) aiming at reducing the adverse pressure gradient over the back part of the section and at the same time maintaining high design lift. Another, lower lift, possibility is to have rounded trailing edge which would also facilitate the manufacturing of the blade especially if the shape is convex.

Flat-back airfoils have been investigated both computationally and experimentally [1-4] and existing wind tunnel and numerical data are used for validating the numerical simulations presented. The most important feature of this type of airfoils is the generation of a von Karman street like wake.
which is expected to excite fluctuations of the aerodynamic loads. In this sense the simulations need to
be unsteady. The work presented is based on Unsteady (U)RANS CFD simulations.

Anticipating the significant challenge this type of analysis involves, two (U)RANS finite volume
solvers and two eddy viscosity models are used for cross checking purposes especially in terms of the
dynamic characteristics of the relevant flows that are not yet known from tests. Aiming at assessing
the eventual effect of compressibility in the CFD simulations, the first solver is an incompressible
solver while the second is a compressible code with low Ma preconditioning. Then in terms of
turbulence closure, two eddy viscosity models are considered: the Spallart-Allmaras (SA) and the k-ω
SST. Both models are widely used and regarded appropriate for aerodynamic flows.

The cases considered concern two modified 35% thick airfoils: the FB 3500 1750 airfoil with sharp
corners at the trailing edge and the NACA0035-1750 with a rounded trailing edge. Both airfoils end
with a 17.5% thick trailing edge. The present work aims at: assessing the capabilities of standard CFD
in predicting this type of flows; detecting the aspects in CFD that require further consideration; and
investigating possible advantages elliptically shaped airfoils could present.

2 Description of the CFD solvers
The first of the two (U)RANS solvers that are used, is an implicit pressure correction solver here
referenced as “incompressible”. The method introduces a matrix-free algorithm for pressure updating
which maintains the compatibility of the velocity and pressure field corrections allowing for
practically unlimited large time steps within the time integration process. Spatial discretization is
performed on a computational domain resulting from a body fitted coordinate transformation using
finite volume schemes. The velocity is stored at grid nodes while the pressure is computed at mid-
cells. This staggering technique allows for pressure field computation without any explicit need of
pressure boundary conditions. A linear 4th order pressure dissipation term is added into the continuity
equation to prevent velocity-pressure decoupling. Further details can be found in [5]. For turbulent
flow computation the standard k-omega (Wilcox) or the k-omega SST [6] is employed. In the present
paper all results were produced with the k-omega SST.

The second is a multi-block MPI enabled compressible solver equipped with preconditioning in
regions of low Ma flow, here referenced as “compressible” [7]. The discretization scheme is cell
centered and makes use of the Roe approximate Riemann solver for the convective fluxes. In space the
scheme is 2nd order accurate defined for unstructured grids and applies the Venkatakrishnan’s limiter
[8]. Also in time the scheme is second order and implicit introducing dual time stepping for facilitating
convergence. The solver is equipped with the Spalart-Allmaras (SA) [9] and the \( k-\omega \) SST [6] eddy
viscosity turbulence models.

3 Results and discussion
3.1 Numerical parameters
O type structured grids have been produced using ANSYS ICEM (see Figure 1). Three grids with
56000, 113000 and 226576 cells were initially tested, all extending to 20c, with 572 points on the
airfoil surface, and \( y^+ \approx 0.1 \) near the wall. Also three time steps were tested: 1, 2, 4 \( 10^3 \). The time
averaged lift \( \bar{C}_l \) was found practically insensitive to the grid changes while for the time averaged drag
\( \bar{C}_d \) grid refinement improved the quality from 8% to 3%. Then as regards the time step the differences
when switching from 4 to 2 \( 10^3 \) were in the order of 15% while the next time step refinement gave
differences of 5%. Based on that the grid of 113000 cells and the time step of 2 \( 10^3 \) were retained for
the rest of the investigation. Clearly a more detailed grid and time step dependency study is needed
including the full range of angles of attack.

3.2 Analysis of the FB- 3500 1750 airfoil
Initially simulations in fully turbulent and fixed transition mode were performed. Trip points were
placed according to the specifications given in [1]: 2% on the suction side and 5% on the pressure
side. The time averaged lift and drag coefficients are shown in Figure 2. Information on the unsteady characteristics of the simulations is given in Table 1.

Up to 15°, in all simulations the flow is attached (at 0° a small separation bubble appears over the concave part of the pressure side). The full attachment of the flow is confirmed by the almost linear increase in lift but also from the skin friction distributions. For the same range of angles of attack, i.e. up to 15°, drag remains almost constant with its lower value appearing at 15°. Compared to measurements, in attached conditions the lift is well predicted while drag is significantly over-estimated. Failing to predict separation at 15° leads to big differences. In fact the measured data indicate a slight slope decrease in the lift curve from 10 to 15° suggesting that separation could have been initiated. The two k-w based CFD sets of simulations agree fairly well especially in terms of averaged drag. The slope of the linear part of the lift curve is lower in the incompressible k-w SST results as compared to those of the compressible k-w SST solver. In between the two lay the results obtained with SA model. Also the measured data are within the range of the CFD results being closer to the incompressible k-w SST predictions. As already mentioned the drag is over-predicted. The k-w results give an over-prediction in attached flow conditions of ~75% while the SA results give double drag. All CFD results agree in predicting a small decrease in drag at maximum lift, i.e. at 15°. Since drag is controlled over this range of angles of attack by the pressure established over the flat part at the trailing edge region, the origin of the aforementioned over-prediction could be linked to either poor resolution within this area or difference in the way drag measurement is carried out.

Measurements at angles of attack >15° are not available and therefore the analysis is based on CFD results comparison. In all CFD solutions, lift drops. The compressible results are similar differing from that of the incompressible ones by ~50%. Worth noticing is that the slope of this drop in the incompressible results agrees very well to the slope of the drop in the measurements which however takes place 5° earlier. Similar remarks can be made for the drag. Once separation over the suction side occurs in the simulations the drag increases significantly. In all CFD results the increase is similar while the slope of the predicted drag curve agrees well with the measured data.

Coming to the dynamic characteristics of aerodynamic load signals, all CFD results give a Str number in the range of 1-1.3 prior to separation but very different amplitudes. FFT analysis indicates that there is also excitation of the 2nd harmonic. The corresponding energy is much smaller and becomes visible only in the CD signals. Also worth noticing is that once separation takes place, i.e. at 20°, the frequency is approximately divided by 2 at values that suggest bluff body behavior. The compressible solver, regardless the turbulence model gives ~10% larger frequency in attached flow conditions as compared to the incompressible one, while when the flow separates the difference is higher. In terms of amplitudes, the incompressible solver gives significantly lower values as compared...
to those of the compressible solver. Tests were made at lower free stream Ma numbers up to 0.01, but the amplitudes in the lift and drag signals did not change. Lack of experimental or numerical evidence on dynamic characteristics of the signals does not allow drawing a conclusion on this issue.

**Figure 2:** Time averaged lift and drag results for the FB 3500-1750 airfoil at various angles of attack. The conditions for all sets are: Re=660000, Ma=0.1, fixed transition.

| $\alpha^0$ | Data set | $\bar{C_L}$ | $\Delta C_L$ | $\bar{C_D}$ | $\Delta C_D$ | Str |
|------------|----------|-------------|--------------|-------------|--------------|-----|
| 0          | Incompressible SST | 0.35            | 0.066        | 0.17 | 0.033 | 1.20 |
|            | Compressible SST    | 0.38            | 0.290        | 0.17 | 0.023 | 1.30 |
|            | Compressible SA     | 0.37            | 0.330        | 0.20 | 0.039 | 1.32 |
|            | Experiment          | 0.42            | -            | 0.098 | - | - |
|            | min/max error       | 9-16%           | 73-104%      | - | - | - |
| 5          | Incompressible SST  | 0.94            | 0.061        | 0.17 | 0.037 | 1.16 |
|            | Compressible SST    | 1.06            | 0.270        | 0.17 | 0.043 | 1.24 |
|            | Compressible SA     | 1.00            | 0.300        | 0.20 | 0.055 | 1.29 |
|            | Experiment          | 1.02            | 0.098        | - | - | - |
|            | min/max error       | 3-7%            | 73-104%      | - | - | - |
| 10         | Incompressible SST  | 1.51            | 0.057        | 0.17 | 0.037 | 1.11 |
|            | Compressible SST    | 1.71            | 0.230        | 0.17 | 0.053 | 1.28 |
|            | Compressible SA     | 1.64            | 0.260        | 0.21 | 0.066 | 1.27 |
|            | Experiment          | 1.51            | 0.24         | - | - | - |
|            | min/max error       | 0-13%           | 12.5-29%     | - | - | - |
| 15         | Incompressible SST  | 2.09            | 0.055        | 0.16 | 0.030 | 1.07 |
|            | Compressible SST    | 2.29            | 0.130        | 0.14 | 0.037 | 1.20 |
|            | Compressible SA     | 2.25            | 0.230        | 0.18 | 0.060 | 1.23 |
|            | Experiment          | 1.03            | 0.38         | - | - | - |
|            | min/max error       | 89-122%         | 55-71%       | - | - | - |
| 20         | Incompressible SST  | 1.56            | 0.20         | 0.37 | 0.12  | 0.51 |
|            | Compressible SST    | 1.99            | 0.29         | 0.31 | 0.037 | 0.70 |
|            | Compressible SA     | 1.97            | 0.37         | 0.35 | 0.047 | 0.68 |
In Figure 3 the pressure and skin friction distributions over one period at 5° are compared between the incompressible and compressible simulations both using the k-w SST model. The mean suction peak is the same and the overall layout of the Cp curves is similar which explains the small difference in the predicted averaged lift. However the incompressible solver produces fluctuating pressure distributions contrary to what the compressible solver provides. This difference is reflected consistently on the corresponding amplitudes. Also similar is the pressure level over the flap back region at the TE which when time averaged will determine the averaged drag value.

![Pressure and Skin Friction Distributions](image)

**Figure 3:** Pressure (left column) and skin friction (right column) distributions at 5°. Upper row: Incompressible k-w SST, Lower row: Compressible k-w SST.

In Figure 4 snapshots of vorticity contours are shown over one period at 5°. The formation of a von Karman street is clearly seen. The sharp corners at the trailing edge force the boundary layer to release vorticity in the free flow exactly at the corners. The asymmetry of the flow leads to an alternating emission of coherent vortices of opposite sign. This fact will lead to load fluctuations at a specific Str number. The differences in amplitude and Str number between the compressible and incompressible simulations is due to the intensity of the vortices emitted as well as the way the generated pressure fluctuations propagate. In the incompressible solver, the propagation takes place at infinite while in the compressible at finite.
Figure 4: Snapshots of vorticity contours over the flat back region at 5° (Incompressible, k-w SST)
Figure 5: U velocity contours and selected streamlines at 20° (Compressible k-w SST). Negative values indicate the separation bubble.

In Figure 5 a different aspect of the flow is shown. At 20° the flow is separated, the separation point being at 31% of the suction side. These snapshots indicate that while the separation point does not change in time significantly, the separation bubble is changing shape and volume resulting load fluctuations. The vortex that at lower angles of attack is shed from the upper corner is now merging with the detached vorticity from the upper boundary layer into an extended a structure with two peaks. This could explain the change in Str number.

3.3 Analysis of the NACA0035-1750 airfoil
The NACA0035 airfoil has been modified in the same way as FB3500-1750 was produced except that the trailing edge is now smooth. Only numerical results are presented using the compressible solver equipped with the k-w SST model. Comparisons are made with the results obtained for the original NACA0035 profile. There was no specific reason for choosing NACA0035, besides the aim of having the same thickness. Otherwise, the specific airfoil differs from the original airfoil that generated The FB3500-1750 in having a much lower design lift.

Time averaged lift and drag results are shown in Figure 6. Error bars indicate the amplitude in the signal. Keeping a substantial thickness for most of the chord length results an important increase in lift. The maximum lift is obtained at 10° and is ~4 times bigger compared to the lift of the original airfoil at the specific angle of attack. However it is half of that FB3500-1750 achieves. The flow over the modified airfoil, remains attached up to 10° as the linear increase in lift indicates. Over this range of angles of attack, vorticity is released at the trailing edge region as in the FB3500-1750 airfoil. The point of release as given by the skin friction surface distribution remains practically unchanged for the specific Re number: 93% and 94% at the suction and pressure side respectively. Separation on the airfoil surface is first detected at 15°. At this angle of attack, the point of separation is found at 37% while it further moves at 22% when the angle of attack is 20°. However the decrease in lift is not steep as expected for such thick airfoils. Coming to drag, the simulations indicate an increase which at 0° is ~50% getting almost zero at 10° and 15°. At 20° the modified airfoil has 10% more drag. This indicates that the wake induced effect on the drag of the modified airfoil is similar to the stall induced effect on the original airfoil.

Compared to the FB airfoil, the Str numbers obtained in the load signals are in general higher (Table 2). However at 20° the values get close which indicates as previously bluff body behaviour. Then as regards the amplitudes, in attached flow conditions they are significantly lower for the NACA 0035-1750 as compared to those of the FB airfoil. Looking into the structure of the wake (Figure 7) it is noted that contrary to the FB airfoil, the separated boundary layers at the sides of the trailing edge do not form well distinct vortices but rather retain their character as shear layers. There is however a
wavy pattern in the wake which eventually gets unstable in higher Re numbers allowing the formation of concise vortex structures.

![Figure 6](image6.png)

**Figure 6**: Time averaged lift and drag results for the NACA0035 with shard and rounded trailing edge. Re=660000, Ma=0.1, fully turbulent.

![Figure 7](image7.png)

**Figure 7**: Snapshots of vorticity contours for the NACA0035-1750 airfoil at 0°, 5° (upper row) and 10°, 15° deg (lower row). A limited range of contour values has been chosen in order to clearly show the extent of the shear layers produced.
Table 2: Comparison of the Str numbers of the load signals at various angles of attack

|          | 0    | 5    | 10   | 15   | 20   |
|----------|------|------|------|------|------|
| FB 3500-1750 | 1.30 | 1.24 | 1.28 | 1.20 | 0.70 |
| NC 0035-1750 | 1.70 | 1.67 | 1.58 | 1.04 | 0.67 |

Another aspect of the difference between the two airfoils is shown in Figure 8. Snap-shots of the manometric pressure in the trailing edge region at $5^\circ$ are compared. There is approximately an order of magnitude difference. This explains the significantly lower load fluctuations in the case of smooth trailing edge and is related to the fact that there are no concise vortex structures generated in the wake. In order to conclude whether this is a characteristic of airfoils with rounded trailing edge, higher Re numbers must be considered as well as airfoils with higher design lift.

![Figure 8: Manometric pressure contours in the wake of the two airfoils at $5^\circ$.](image)

4 Conclusions

Two 35% thick airfoils, both having a 17.5% thick trailing edge were analyzed on the basis of CFD simulations: the first with sharp corners and the second with a rounded finishing.

1. **CFD vs measurements**: Averaged lift is well predicted until separation takes place. CFD predicts separation with a $5^\circ$ delay but the drop in lift follows the trend of the measured data especially in the incompressible CFD results. The constant drag in attached conditions and the slope of the increase in drag once separation takes place are reproduced. However the drag level in attached conditions is overestimated. The delay in predicting separation could be due to 3D effects while the level difference in drag could be related to the measuring procedure.

2. **Incompressible vs compressible CFD**: Simulations based on the k-ω SST model give similar time averaged lift and drag values. Also the Str number of the loading response to vortex shedding is similar. However large differences in load amplitude have been found and associated to the different time evolution of the pressure distribution on the airfoil surface. A possible explanation is that pressure fluctuations propagate differently in the two formulations.

3. **K-ω SST vs SA**: The two models have produced similar results. A level difference in drag is however noted. The implementation of the two models was such that transition from laminar to turbulent flow took place exactly at the defined transition point. A more flexible transition modelling should be included in future work. Due to the inherent dynamics produced in this type of flows 3D as well as DES or LES turbulence modelling should be considered.
4. **Challenges for CFD**: The significantly different amplitudes in the lift and drag responses as well as the importance of these load fluctuations to the design of blades, makes it necessary to cross check the present findings with unsteady measurements and other numerical results. Another important point concerns the delay in predicting separation. This calls for: a 3D analysis so as to see whether a nominally 2D flow switches to 3D as a result of spanwise instabilities.

5. **Design implications**: Non-conventional airfoils either having a sharp cornered TE or a rounded one increase lift substantially [10]. Drag is also increased, but the level at which this happens is not clear (for the flat-back CFD over-predicts the measured time averaged value, while for the rounded airfoil, the recorded increase is not big). Also of particular importance are both the frequency and the amplitude in the response signals of lift and drag due to their link to aeroelasticity. In the particular airfoils, load fluctuations on the rounded were found significantly lower suggesting the need for a different shape optimisation.

The whole analysis was carried out at a low Re for the sake of comparison with measured data. An analysis of the aerodynamic performance at higher Re numbers corresponding to wind turbine operational conditions is needed before concluding on the design implications these type of airfoils would have.

**References**

[1] Baker JP, van Dam CP, Gilbert LB, 2008, "Flatback Airfoil Wind Tunnel Experiment", SANDIA report SAND2008-2008

[2] van Dam CP, Mayda EA, Chao DD, 2008, "Computational Design and Analysis of Flatback Wind Tunnel Experiment", SANDIA report SAND2008-1782.

[3] Baker JP, van Dam CP, 2008, “Drag reduction of blunt trailing-edge airfoils”, BBAA VI International Colloquium on: Bluff Bodies Aerodynamics & Applications, Milano, Italy, July, 20–24 2008

[4] Sorensen NN, 2011, "A Small Study of Flatback Airfoils", in: *Presentations from the Aeroelastic Workshop – latest results from AeroOpt*, Morten Hartvig Hansen (Ed.) Risø-R-1769(EN), Risoe-DTU, 2011

[5] Chaviaropoulos P, 1994, ”A Matrix-Free Pressure Solver for the Incompressible Navier-Stokes Equations Application to 2-D Flows”, Proceedings of ECCOMAS Conf., Stuttgart, Germany

[6] Menter FR, 1994, "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications", *AIAA Journal*, Vol. 32, No. 8, pp. 1598-1605.

[7] Papadakis G, 2011, “Formulation of a cell-centered (U)RANS compressible solver”, PhD Report, NTUA

[8] Venkatakrishnan V, 1993, "On the Accuracy of Limiters and Convergence to Steady State Solutions", AIAA Paper 93-0880.

[9] Spalart PR, Allmaras SR, 1994, "A One-Equation Turbulence Model for Aerodynamic Flows" *Recherche Aérodynamique*, No. 1, pp. 5-21.

[10] Jackson KJ, Zudeck MD, van Dam CP, Standish KJ, Berry D, 2005 “Innovative design approaches for large wind turbine blades”, *Wind Energy*, 8, 141-171
Figure 9: Snapshots of vorticity contours over the flat back region at 20°