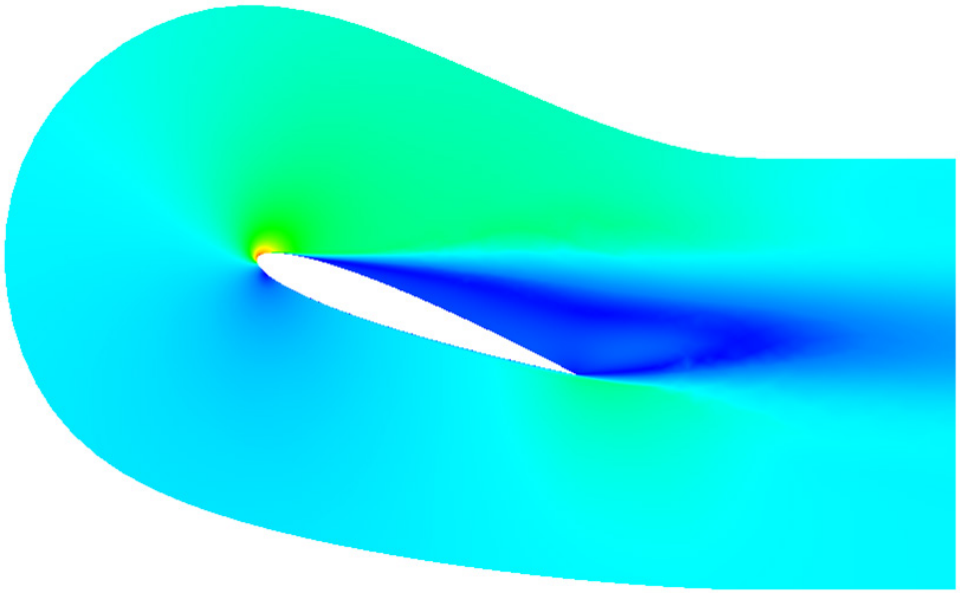
Autodesk Simulation CFD External Flow Validation: NACA 0012 Airfoil



Note: The following results were presented in a webinar as part of the Autodesk Build Your Simulation IQ series. This webinar can be found at YouTube in the 'AutodeskSim360' channel titled "Review NACA0012 2D Airfoil Model in Autodesk Simulation CFD".

Introduction

Airfoils have been studied and used for over a century. Their applications – and thus their design – have varied widely over their history. Yet despite their fundamental mainstay in engineering, the ability to simulate airflow around an airfoil has – for the most part – remained elusive. Accurate results have been extremely difficult to achieve, leaving designers to rely on historical test data and trial and error.

But simulation technology continues to make strides – making it not only possible to analyze lift and drag, but also practical. The following paper details a recent validation study examining the ability of Autodesk® Simulation CFD to conduct a 2D simulation of airflow around a standard NACA airfoil and predict lift and drag. Results were compared against published test data and demonstrate the level of accuracy that can be achieved using a straightforward modeling approach and just a few additional solver controls available to enhance the modeling.

Background: What are the NACA Airfoils?

NACA stands for the National Advisory Committee for Aeronautics. It was a federal agency developed in the early 20th century to cultivate aeronautical research. With the rise of the space age, it was dissolved and transformed into NASA.

In the early days of flight, NACA began to observe key relationships in successful airfoil design. They developed equations that could utilize these relationships to generate a consistent family of airfoil shapes. The NACA 0012 is part of the four-digit series. The digits in the name are parameters that are entered into the equations to precisely generate the cross-section of the airfoil. The numbers are designated as follows:

- The first digit describes the maximum camber as a percentage of the chord length.
- The second digit describes the distance of the maximum camber from the airfoil leading edge in tenths of a chord
- The last two digits describe the maximum thickness of the airfoil as a percentage of the chord.

Airfoils with a series number beginning with 00 – such as the NACA 0012 - are symmetrical and have no camber.

The equation for the NACA 0012 airfoil is given by:

$$y_t = 5tc\left[0.2969\sqrt{\frac{x}{c}} + (-0.1260)\left(\frac{x}{c}\right) + (-0.3516)\left(\frac{x}{c}\right)^2 + 0.2843\left(\frac{x}{c}\right)^3 + (-0.1015)\left(\frac{x}{c}\right)^4\right]$$

Where
c: Chord length,
x: Position along the chord from 0 to c
y: Half thickness at a given value of x
t: Maximum thickness as a fraction of the chord (XX/100)

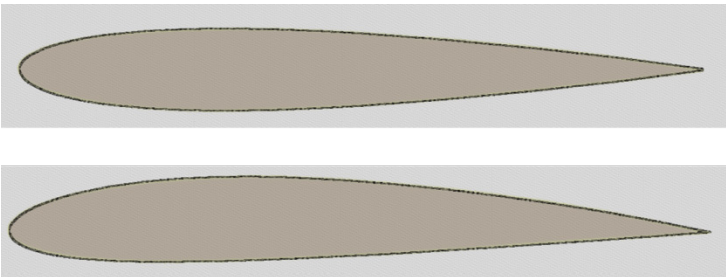


Figure 1. Example NACA Airfoil

The NACA airfoils have since been used for validation cases for turbulence models. Many NACA airfoils have been physically tested and have extensive data use in evaluation of advanced Computational Fluid Dynamics codes. The following study compares Simulation CFD results for lift and drag against two sets of test data for the NACA 0012 airfoil, one of the most tested of the airfoils [McCroskey].

Experimental Data

Data Sources

Two sets of data were used for comparison. As detailed below, the data included testing of the airfoil under multiple angles of attack as well as different surface conditions. This allowed for evaluation of simulation prediction of stall under both laminar and turbulent flow conditions.

1. *Effects of Independent Variation of Mach and Reynolds Numbers on the Low-Speed Aerodynamic Characteristics of the NACA 0012 Airfoil Section* – **Charles Ladson** (NASA Langley 1988) [(1)]

   This NASA Langley test set looked at NACA 0012 for various operating conditions and angles of attack. A particular note about this testing is they looked at a tripped vs. un-tripped surface. In other words, in the leading 5% of the airfoil, they applied sand paper of different roughness (60 and 80-grit in this case) which served to create a layer of turbulence on the leading edge. Otherwise, flow is purely laminar on the leading edge of this airfoil geometry. The testing revealed that excellent correlation could be achieved for drag when comparing experimental and theory when using the fixed transition or tripped airfoil.

   Test Conditions
   Mach: 0.05 – 0.36
   Reynolds: 2 – 12E6
   AOA: 0 – Max Lift
   Surface Conditions:
   - Smooth
   - Tripped (60-grit)
   - Tripped (80-grit)

2. *Aerodynamic Characteristics of Seven Symmetrical Airfoil Sections Through 180-Degree Angle of Attack for Use in Aerodynamic Analysis of Vertical Axis Wind Turbines* – **Robert Sheldahl** (Sandia 1981) [(2)]

   These tests, conducted by Sandia National Laboratories, looked at multiple airfoils for application in vertical axis wind turbines. This included a range of Reynolds numbers and a full rotation of angles of attack (0-180 deg).

   Test Conditions
   7 Airfoils Sections
   Reynolds: 10E4 – 10E7
   AOA: 0 – 180°
   Surface Condition: Smooth

The combined experimental data set for this validation study consists of tests at Reynolds Number around 2 x10^6 including two tests of smooth surface data and two sets of tripped (60-grit and 80-grit) surface data. The resulting plots of experimental lift coefficient and drag coefficient are shown below.

Sheldahl: Re = 2 x 10^6

Ladson: Re = 1.76 x 10^6

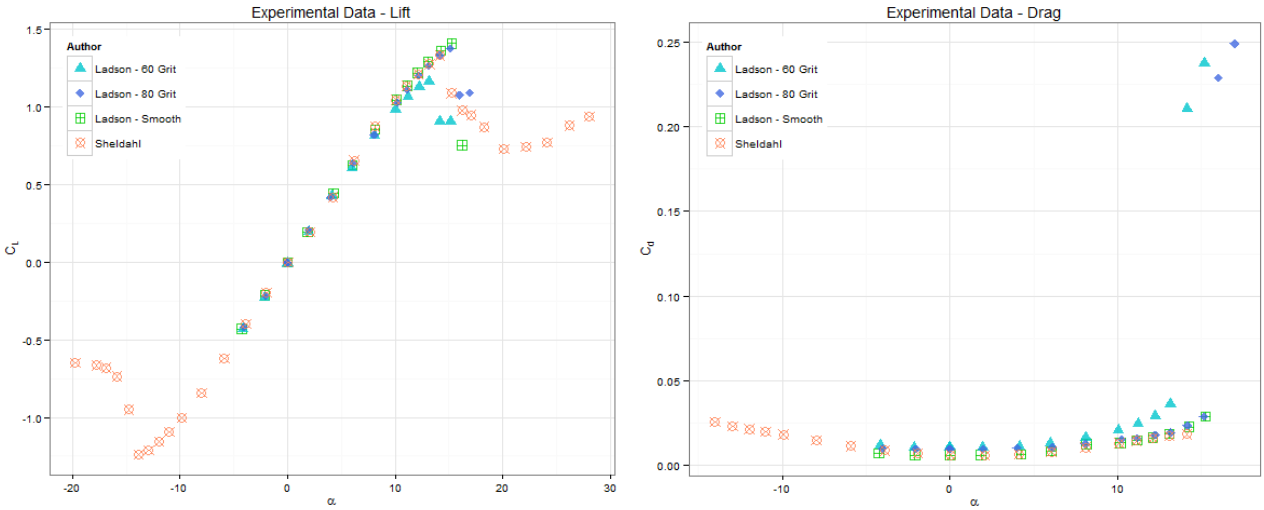


Figure 2. NACA 0012 Experimental Data for Lift (left) and Drag (right)

Observations of Experimental Data

Upon examining the experimental data above, the following is observed. All measured lift results, especially at lower angles of attack, correlate very well but start to separate at larger angles of attack. Smooth results show extra lift before stall followed by much greater spread and more uncertainty.

Measured drag shows a more noticeable difference between smooth results and tripped results at lower angles of attack. It is noticed that the tripped results seem to diverge at larger angles of attack, with the 80-grit results actually converging with the smooth results at larger angles of attack

Assessment of Wind Tunnel Results for the NACA 0012 Airfoil

The variability shown above raises the question: how reliable is the test data and which test values should be used for validation purposes?

It should be recognized that two-dimensional experiments are extremely difficult to achieve. The differences seen above occur largely at higher angles of attack near stall where the experiment is likely no longer two-dimensional. Because of this difficulty, work at NASA Ames in 1987 looked at the results from over 40 different wind tunnels to compare tests and quality of results. The results were published in a paper entitled “A Critical Assessment of Wind Tunnel Results for the NACA 0012 Airfoil” [(3)]. The study showed a range of values and contributed some key findings.

The following shows a composite of results for NACA 0012 at zero angle of attack (AOA). Figure 3 shows lift-curve slope data in a limited Re range for tests deemed to stand out for “most nearly eliminating the important sources of wind-tunnel errors”. It is noted that most of the results in this group have no trip. It was found that a good fit of the data is given by:

$$\beta C_{l_\alpha} = 0.1025 + 0.00485 \log(\mathrm{Re}/10^6)$$

Figure 4 shows results for drag coefficient at AOA = 0°. The set of data selected for the lift-slope results is collected as "Group 1". The curves represent approximate fits for *Group 1 tripped* and *Group 1 un-tripped* data and are given by:

Tripped: $C_{d_0} = 0.0017 + 0.91/(\log(\mathrm{Re}))^{2.58}$

Un-Tripped: $C_{d_0} = 0.0044 + 0.018\mathrm{Re}^{-0.15}$

Group 2 is comprised of tests where data “generally agree with both the lift and drag criteria (expressed in the fit equations) to within +/-0.0040 for slope and to within +/- 0.0010 for Cd¬0.”

In summary, test data that falls along the curve fits for Group 1 was determined to provide the most reliable results. **Thus, for purposes of evaluating a CFD solution for predicting lift and drag, we will be looking for simulation outputs to resonably meet those criteria.**

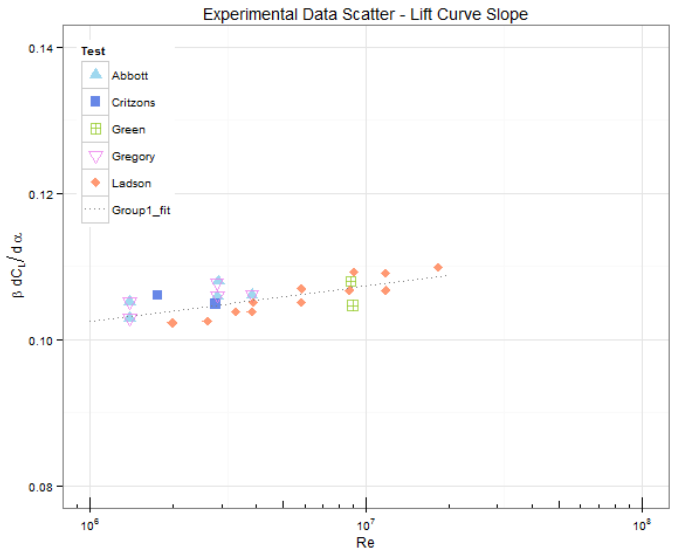


Figure 3. Experimental Data Scatter – Lift Curve Slope

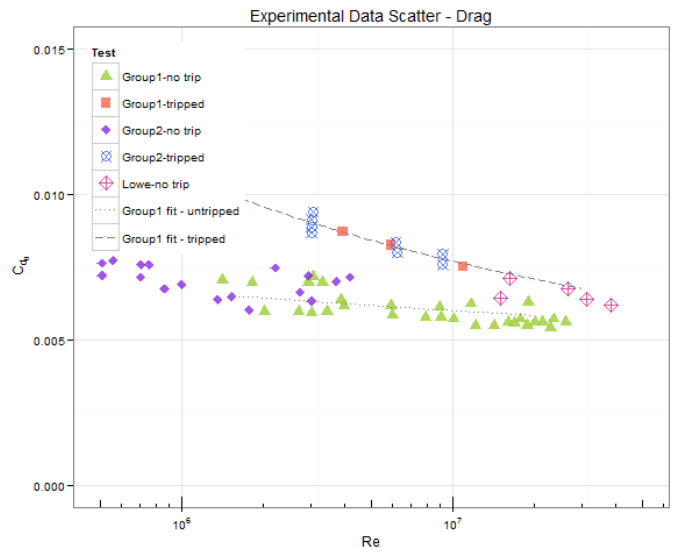


Figure 4. Experimental Data Scatter - Coefficient of Drag

Simulation

A 2D NACA 0012 airfoil with chord length of 1 meter was used for simulation. Wind speed was at approximately 26.56 m/s representing a Reynolds Number of 1.76 x10^6 (corresponding with test data). The Angle of Attack (AOA) was varied incrementally from -4 deg to 20 deg in order to capture stall as well as some reverse rotation.

Simulations were run in Simulation CFD 2015 using standard advanced turbulence techniques with the added use of two specific flag settings available in Simulation CFD to enhance the chosen SST (shear stress transport) k-omega turbulence model. The following sections detail the CFD simulation, including construction of the airfoil model, applied simulation settings and meshing strategy.

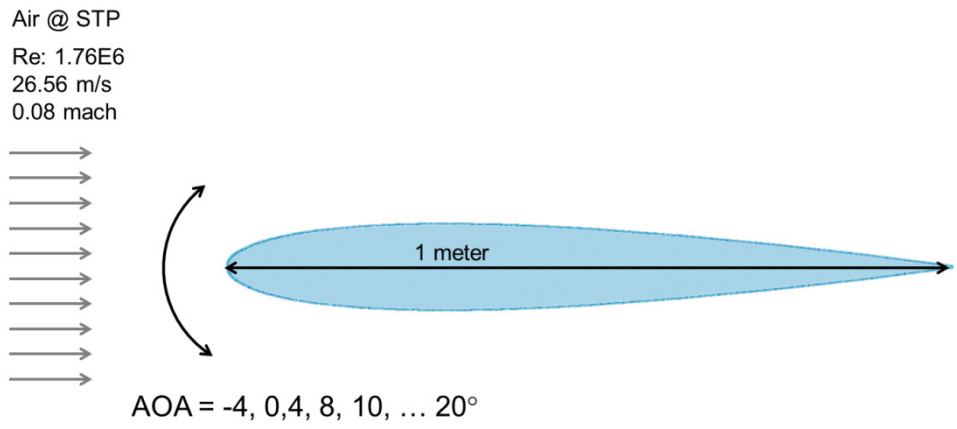


Figure 5. Simulation Summary

Model

The model was created in Autodesk Inventor Professional by simply entering the 2D equation for the cross section. I-logic was utilized to set up Simulation CFD for a parametric study of several different angles of attack. See Figure 6.

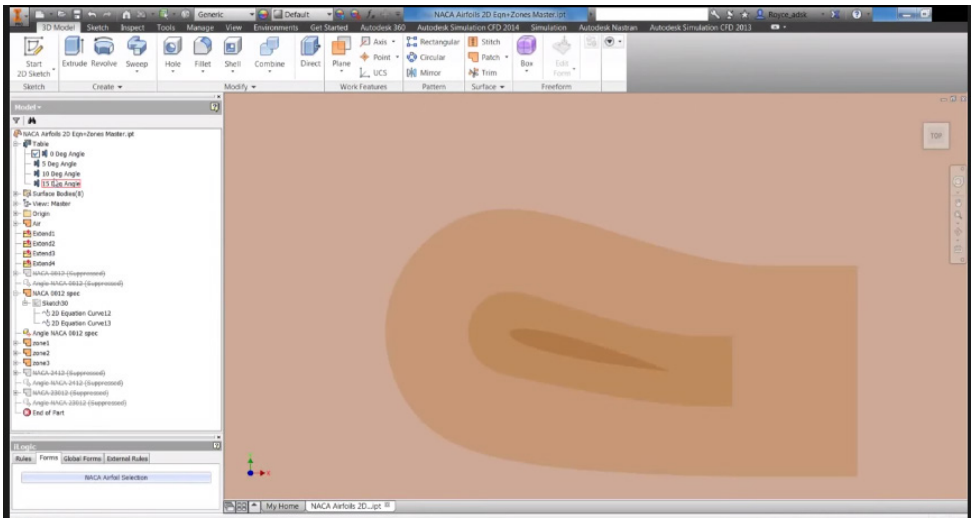


Figure 6. NACA 0012 Model Created in Autodesk Inventor Professional

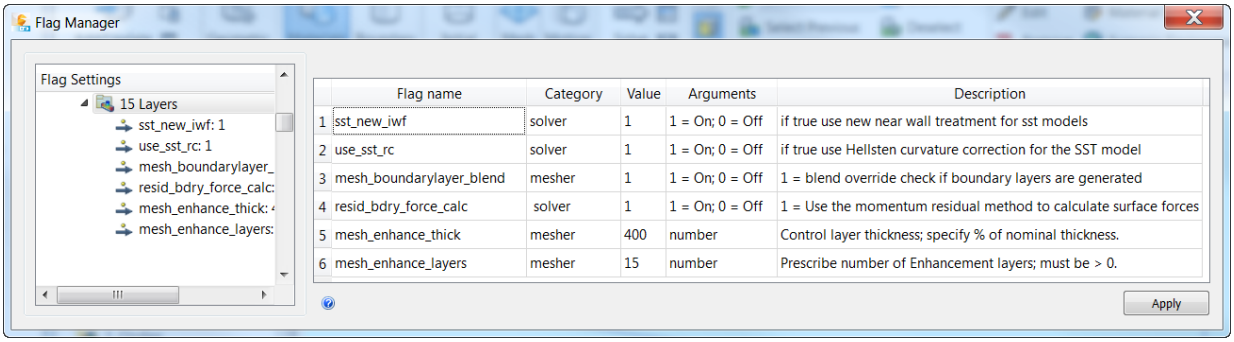
Modification of Airfoil Trailing Tip

The model uses the NACA profile with a small rounded trailing tip instead of a zero thickness tip at the trailing end of the airfoil. Using a zero thickness trailing tip requires modification of the NACA coefficients and results in a slight reduction in angle on the back side due to a numerical geometry closure problem with the true NACA equation coefficients.

Simulation CFD Settings

A few Simulation CFD options were utilized to improve analysis of external aerodynamics in this study. The simulation largely followed a typical set-up technique for advanced turbulence modeling, but a couple additional solver controls were utilized to enhance the SST k-omega turbulence model for the NACA 0012 airfoil.

These options were employed via the following flags for meshing and solver controls - as shown in Figure 7. (These settings were used for all AOA despite different wake activity ensuing at larger angles).



	Flag name	Category	Value	Arguments	Description
1	sst_new_iwf	solver	1	1 = On; 0 = Off	if true use new near wall treatment for sst models
2	use_sst_rc	solver	1	1 = On; 0 = Off	if true use Hellsten curvature correction for the SST model
3	mesh_boundarylayer_blend	mesher	1	1 = On; 0 = Off	1 = blend override check if boundary layers are generated
4	resid_bdry_force_calc	solver	1	1 = On; 0 = Off	1 = Use the momentum residual method to calculate surface forces
5	mesh_enhance_thick	mesher	400	number	Control layer thickness; specify % of nominal thickness.
6	mesh_enhance_layers	mesher	15	number	Prescribe number of Enhancement layers; must be > 0.

Figure 7. Simulation CFD Scenario-Level Flags

Meshing

Accuracy for external aerodynamics analysis requires special considerations of mesh quality along the walls of the airfoil. This includes the following considerations:

- Ensure the flow gradients in the boundary layer are captured within the wall layer elements
- Avoid an abrupt change in mesh height at the transition from prism wall layers to triangular elements
- Highest accuracy is achieved when y+ < 0.3

Three mesher flags were employed; the second two allow adjustment of mesh enhancement beyond what the UI controls allow. The meshing process is described in greater detail in the next section.

mesh_boundarylayer_blend - This flag enables smoother mesh around sharp edges – such as the airfoil – and adds more resolution from the wall mesh into the core. Use this flag to introduce smoother mesh enhancement layers around sharp edges and other outwardly-protruding geometry features.

mesh_enhance_thick – This flag controls the mesh enhancement layer thickness. It's value is a percentage multiplier applied to UI value for layer factor. In this case, it was set to 400 or 400% of layer factor (4x thicker than default thickness).

mesh_enhance_layers - This flag simply sets the default number of layers generated by Mesh Enhancement. It has been used to add more layers for a total of 15. The UI supports up to 15 layers. This option is shown as a demonstration if more layers are needed and provides a means to enforce certain options.

Solver Controls

Several solver flags were used to leverage solver options available for sensitive external aerodynamics analysis like the NACA0012. These included an updated wall force calculator and three advanced turbulence solver controls used to modify the default SST model.

resid_bdry_force_calc 1 – This flag computes the wall forces and properly distributes it from the nodes to the logical CAD surfaces to report in the wall calculator. (This is a newer form of boundary force calculation, considered more accurate for wall forces.)

use_sst_rc 1 (On) – This modifier helps to yield more accurate flow results such as pressure and velocity fields in devices like cyclones and airfoils - specifically when high flow curvature is critical to the accuracy of an analysis.

sst_new_iwf 1 (On) – This flag enables a more accurate method of solving for the pressures and velocities and temperature near no-slip walls in conjunction with the SST turbulence model.

NOTE: Use of the SST Modifiers (previous two flags) is explained in greater detail in Appendix A.

Convergence Controls

In addition to the above enhancements, results were slightly improved with adjustment of convergence controls. Simulation CFD uses several criteria to automatically determine a converged solution and end an analysis. Automatic control is activated by enabling **Intelligent Solution Control** in the **Solution Controls** window. Then **Advanced Solution Control** can then be used to select **Automatic Convergence Assessment** and edit criteria individually. In this case, controls were increased to add an order of magnitude to each control. (Usually 1-2 orders of magnitude are sufficient for sensitive external aerodynamics studies.)

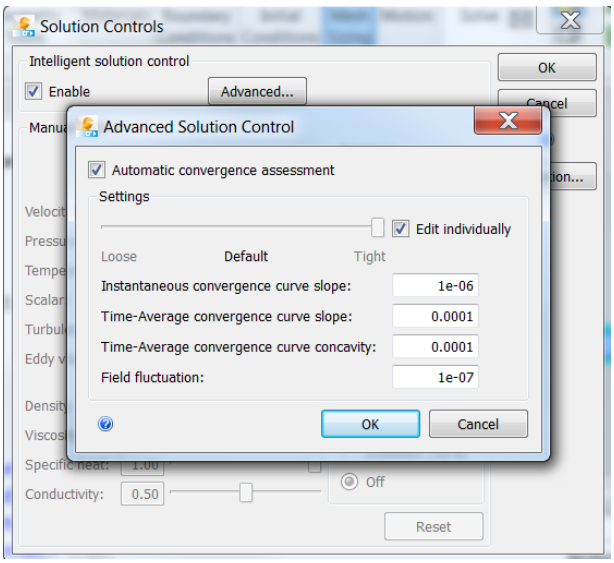


Figure 8. Adjusted convergence controls with 2 orders of magnitude adjustments

Meshing and Boundary/Initial Conditions

The simulation settings described above for meshing center around two goals. First, with a model like the airfoil, it is good to have a gradual transition of element sizes – both within the wall layer and in the transition from wall layer to tetrahedral mesh. The more gradual, the better. Then, the desire is to make sure that the mesh enhancement thickness is thick enough to capture the boundary layer along the walls while maintaining a Y+ value of less than 0.3 for the k-omega SST. Typical refinement regions were incorporated, including two in the wake behind the trailing tip of the airfoil to capture velocity profile leaving that edge. This is shown in the figures below. The mesh scheme is summarized as follows:

- 15 Enhancement Layers
- mesh_enhance_thick (%): 400 (4x layer factor)
- Layer Factor: 0.84 (Effective layer factor is 336)
- Airfoil edge mesh: 2.5 mm
- First Region: 5 mm

Boundary conditions included velocity coming in, slip conditions on top and bottom and a pressure on the outlet. Additionally, an initial velocity condition was applied to the entire domain. This simply starts the air moving and saves some iterations in the simulation

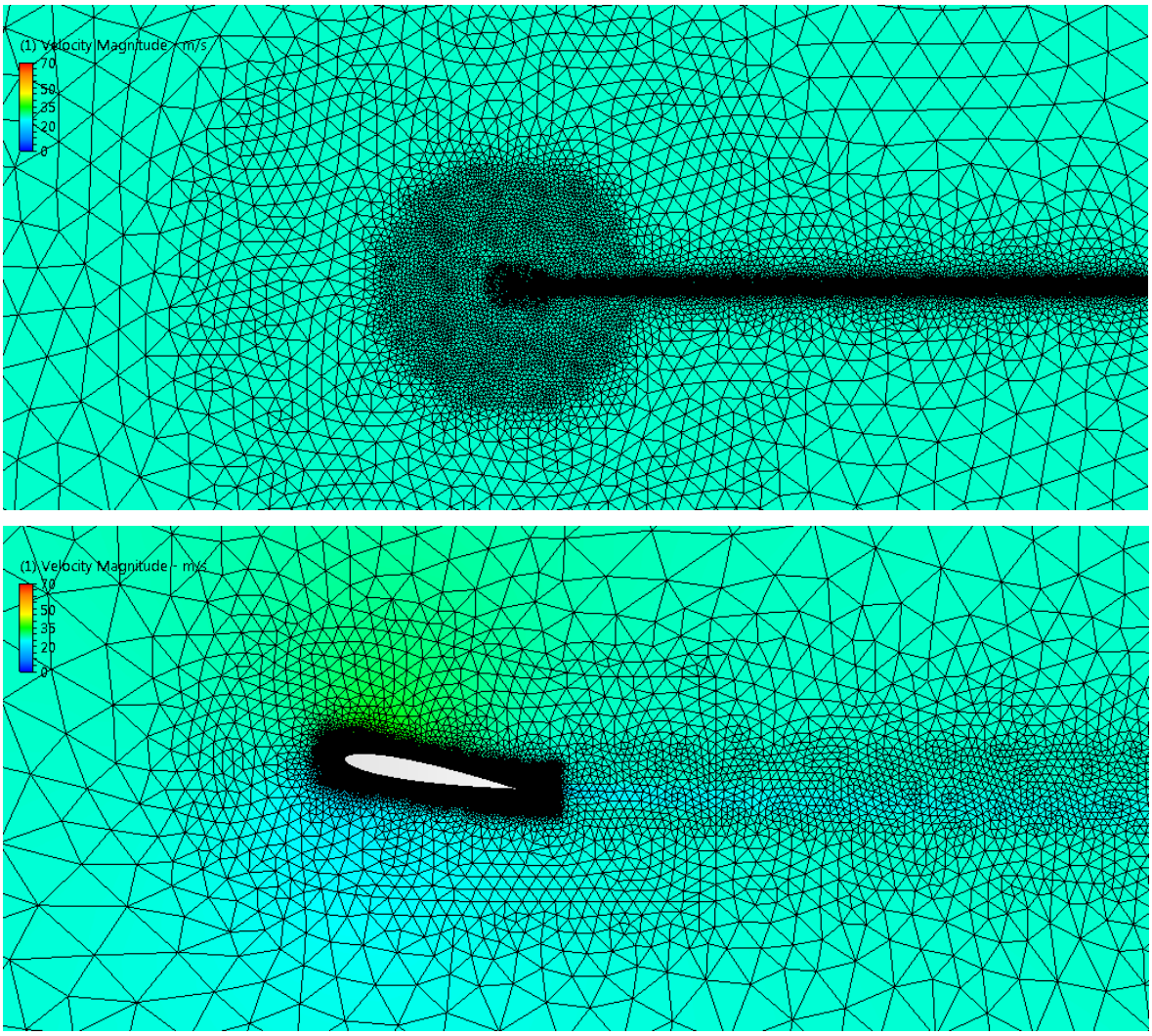


Figure 9. Airfoil mesh refinement regions

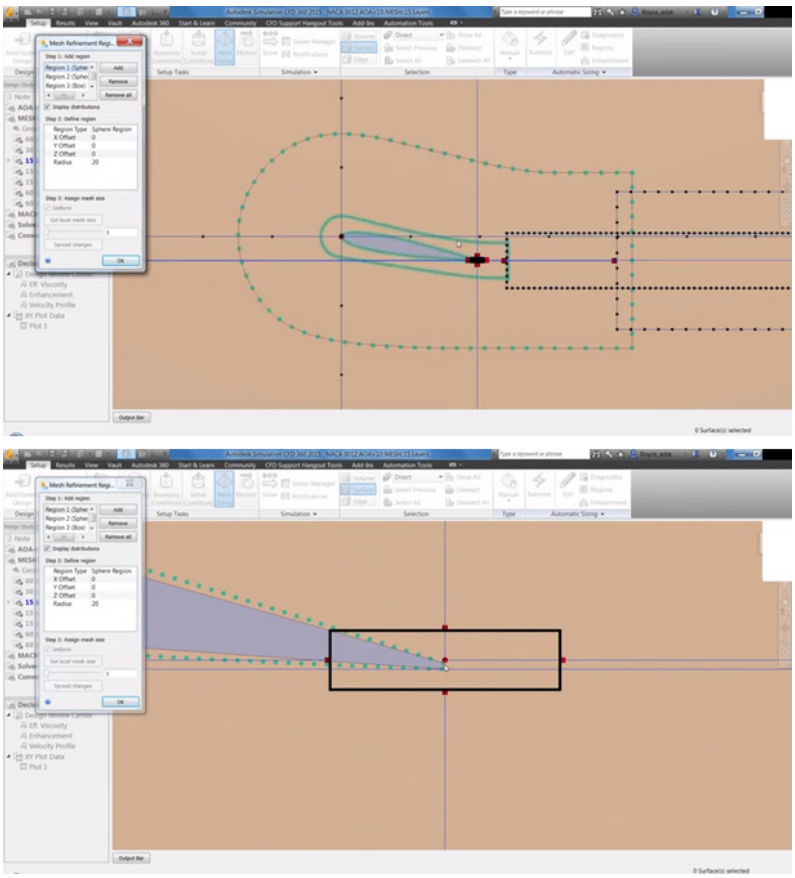


Figure 10. Airfoil trailing tip refinement for wake

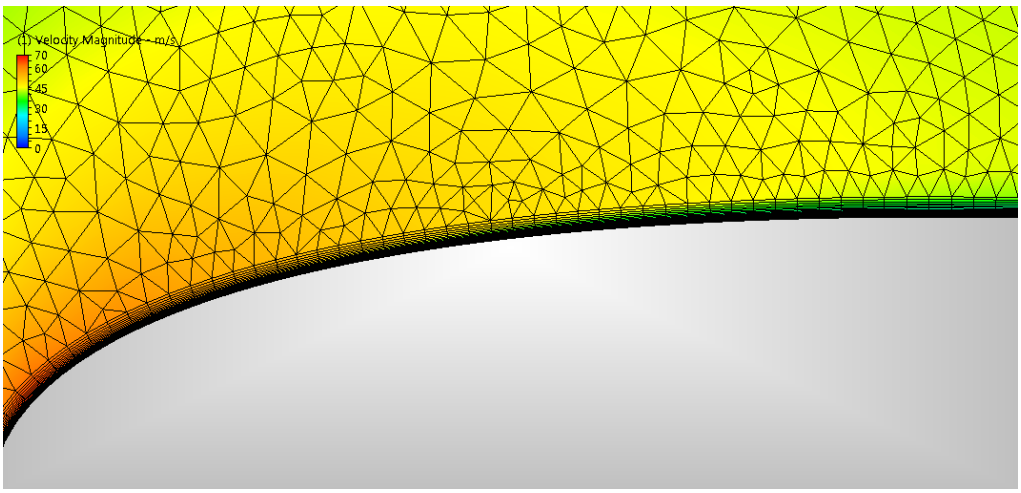


Figure 11. 15 Enhancement layers transition to core mesh

Simulation Results

The results of the simulation described above are shown plotted with the referenced experimental data. It can be seen that results for lift are very good up until stall, capturing peak lift at around 15 deg in correlation with the 80-grit measurement before tapering off. The drag results also correlate well with the tripped results also up until this same point. It is noted here that this correlation has much to do with the SST k-omega turbulence model. The model assumes a minimum amount of turbulence always exists and does not have a means for solving for a full laminar to turbulence transition. This prevents it from approximating the untripped test data.

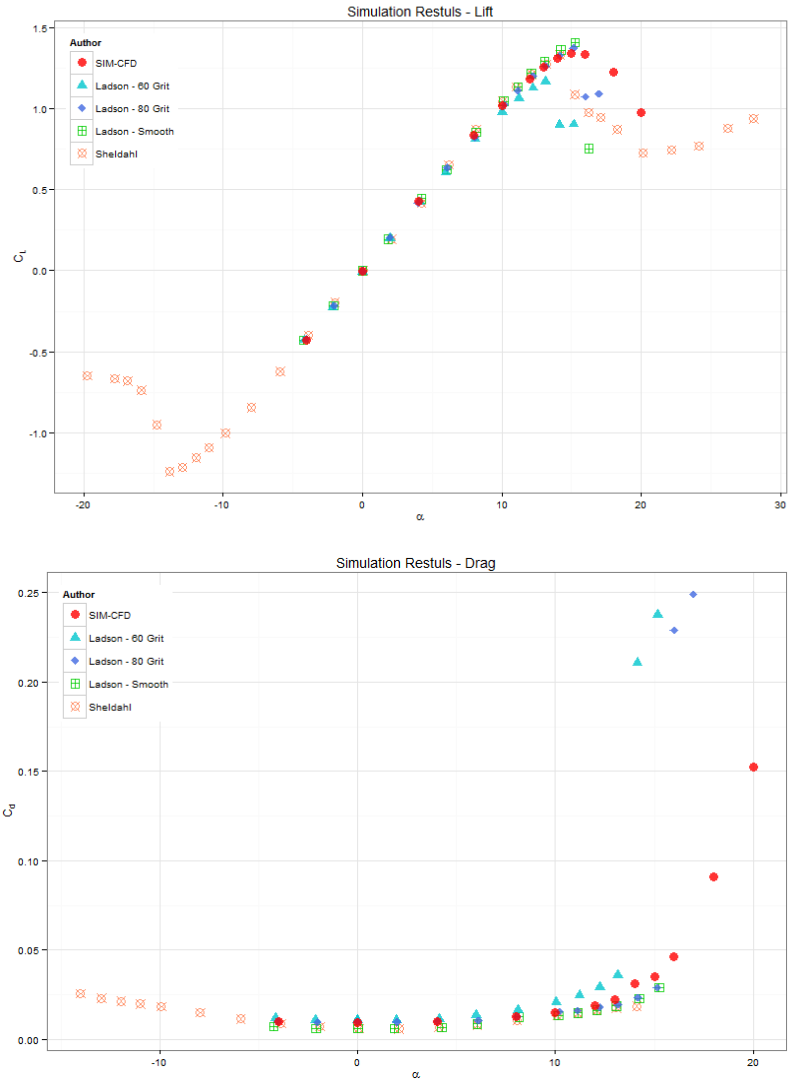


Figure 12. Experiment vs. Simulation Results for Lift (top) and Drag (bottom)

As noted earlier, the experimental data shows some variability and results can be evaluated against best practices (referred to as Group 1). It is useful to see where the Simulation CFD results fall within the lift slope and drag coefficient scatter data shown before. It can be seen below that the simulation results are near the desired curves for Group 1 (most ideal tests) for both lift and for measures of tripped drag.

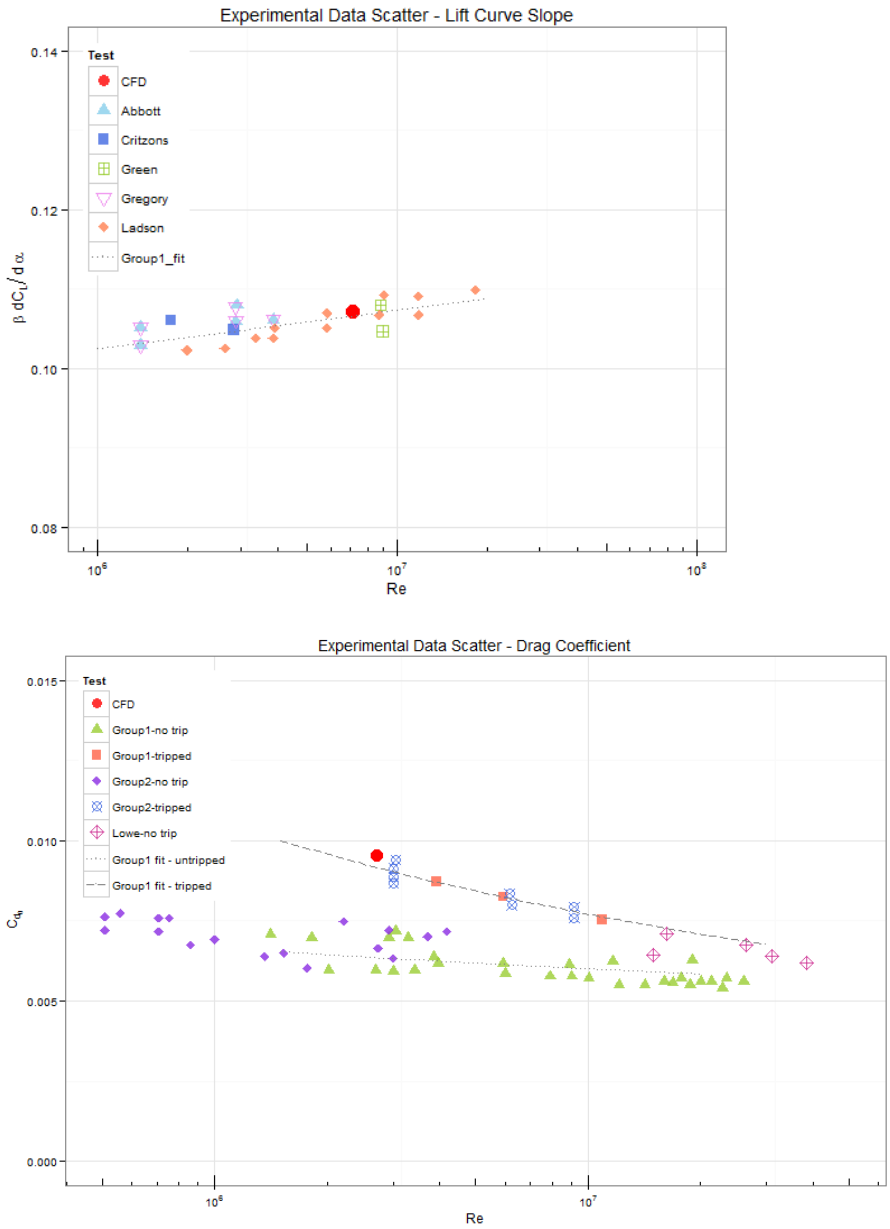


Figure 13. Simulation CFD Results Compared to Experimental Data Scatter for Lift (top) and Drag (bottom)

Conclusions

In this study, Simulation CFD was used to analyze 2D external airflow around a given airfoil and predict lift and drag. Simple meshing enhancements along with advanced turbulence modeling techniques – including enhancements for the SST k-omega turbulence model - were applied to most accurately simulate the airflow and predict performance.

Performance was evaluated against industry-standard test data. Results for lift were very good all the way up to stall, capturing peak lift at 15 deg. For both lift and drag, results correlate well with the experimental results for “tripped” surfaces – particularly the 80-grit – more so than “smooth” surfaces. It was noted that CFD predictions obtained via a turbulence model (such as the SST k-omega applied in this scenario) will inherently correlate more with a surface that prompts turbulent flow rather than laminar.

Results show less correlation after the predicted stall angle. It was noted that at high angles of attack there is increased wake activity and experimental conditions at this point are likely exceeding 2-dimensional flow. However, there is perhaps more that can be done to achieve greater correlation at larger angles of attack – such as including more mesh or looking at oscillations in instances of large wake structures.

Works Cited

1. **Ladson, Charles.** *Effecgts of Independent Variation of Mach and Reynolds Numbers on the Low-Speed Aerodynamic Characteristics of the NACA 0012 Airfoil Section.* s.l. : NASA Langley, 1988. NASA Technical Memorandum 4074.

2. **Shedahl, Robert and Paul, Kilmas.** *Aerodynamic Characteristics of Seven Symetrical Airfoil Sectoin Throug 180-Degree Angle of Attack for Use in Aerodynamic Analysis of Vertical Axis Wind Turbines.* s.l. : Sandia , 1981. SAND80-2114.

3. **McCroskey, W. J.** *A Critical Assessment of Wind Tunnel Results for the NACA 0012 Airfoil.* s.l. : NASA Ames Research Center, 1987. NASA Technical Memorandum 100019.

4. *Numerical Study Comparing RANS and LES Approaches on a Circulation Control Airfoil.* **Rumsey, Christopher L., Nishinio, Takafumi.** 5, s.l. : International Journal of Heat and Fluid Flow, October 2011, Vol. 32, pp. 847-864.

Appendix A - Use of Special Flags Explained

Use of both of the SST modification flags was required to get good correlation with test data.

The **use_sst_rc** flag applies a Hellsten curvature correction factor to the SST k-omega turbulence model. This was applied to address what is known as the Coanda Effect – or the tendency for a fluid jet to be attracted to a nearby surface. This is illustrated in the following figure taken from a paper out of NASA Langley [4]. In Figure 14, test results show how the fluid flowing over the back side of an airfoil has a tendency to pull away or separate from the airfoil. By comparison, the simulations at the right show results with and without SST RC. Without this correction, it can be seen that the fluid does have a tendency stick to the airfoil and pull around to the underside – actually flowing forward. Once the curvature correction is applied, it can be seen that the fluid is predicted to separate from the curved tail. Figure 15shows Simulation CFD results with the curvature correction. It can be seen that the simulation captures the separation and the recirculation region in front of the separation.

Using this option should slightly slow down the time to convergence and most large body external aerodynamics models shouldn't need to use this option. This is why this is off by default.

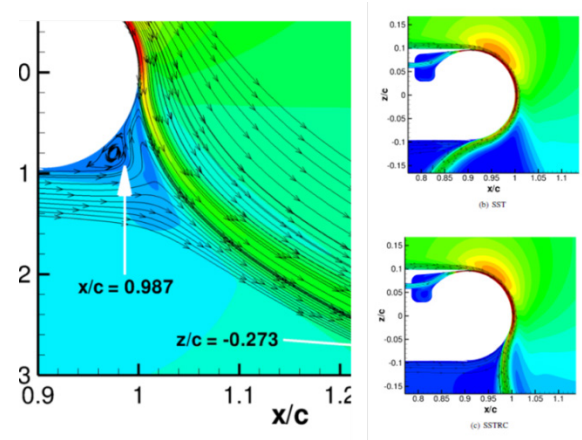


Figure 14. Coanda Effect - Results from NASA Langley[4]

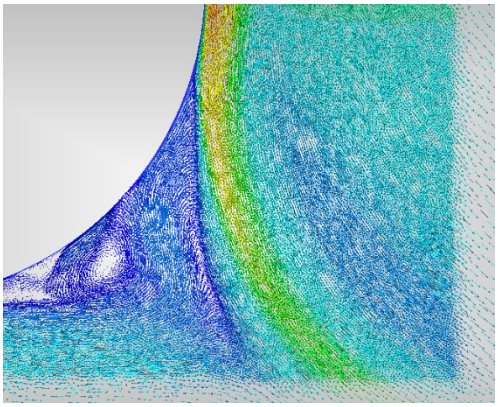


Figure 15. Example effects of use_sst_rc flag

The **sst_new_iwf** flag is used to implement a k-omega SST slip wall formulation. This flag allows for a non-zero velocity at the wall. This method has shown to be a more accurate method of solving for the pressures, velocities and temperature near walls in conjunction with the SST turbulence model.

Note: There is a slip wall formulation for the k-e model as well.