EasyFlowfield Tutorial 7: Vary AOA to Match Lift from Wind Tunnel Experiment

SmartNumerics Simulation Solutions Inc.

Version 6
June 5, 2020

In this exercise you will simulate turbulent flow about an airfoil. You will learn how to specify automated variation of the angle of attack (AOA) until a desired value of lift is reached. A succession of progressively finer grids is used to obtain an optimal value of AOA for a given far-field Mach number. Finally, far-field Mach number is adjusted to get the best fit to the experimental values of pressure along the surface of the airfoil. Details are provided on redoing output with a change in the selected variables and use of ParaView to compute new variables such as the coefficient of friction from existing variables. **Please read the validation document on turbulent flow about airfoils first.**

CFD codes are validated in part by comparing computed airfoil forces, pressure profiles, and friction profiles to data collected using airfoil wing models inserted in wind tunnels. The angle of attack used in the (typically) two-dimensional simulation must be made smaller than the experimental angle of attack to adjust for the effect of the wind tunnel walls. The need to adjust AOA is mostly due to the effect of the top and bottom walls of the tunnel. The tunnel side walls also effect the measurement due to the presence of the side-wall boundary layers. These boundary layers reduce flow near the walls and increase it outside of the boundary layers. The ratio of the wingspan to chord must be large enough so that measurements taken at midspan are not appreciably affected by the side wall boundary layers. The simulation may match experiment more closely if the far-field flow speed in the simulation is slightly increased.

The degree of agreement between the 2D simulation with adjusted AOA and Mach number and the 3D experiment, is used as a measure of the suitability of the turbulence model being used. Different turbulence models require slightly different values of adjusted AOA to obtain the best match to experiment. Once, a satisfactory match is obtained using a suitable turbulence model, various features of the flow field may be examined with an amount of detail not possible with experimental measurements.

The same turbulence model implemented by different authors may require different values of adjusted AOA and flow velocity to match experiment. The turbulence model may be implemented in slightly different ways. The details of the flux calculation including the spatial accuracy and choice of limiter function can affect the drag and thus the optimal AOA. The simulations typically match experimental pressures reasonably well along the underside of the airfoil. With AOA and Mach number adjusted to match experimental shock location, the pressures along the top of the airfoil match experiment reasonably well upstream of the shock except for leading edge where turbulence models generally do not model transition from laminar flow to turbulent flow.

Cook, McDonald, and Firmin [1] performed a series of turbulent tests in a wind tunnel using the RAE2822 airfoil. These tests have been used as the basis of simulations by various workers. See, for example, Holst [2]. The conditions used in the following tutorial are based on transonic case 6 of Cook, McDonald, and Firmin [1]. The specified experimental conditions were Mach 0.725, angle of attack 2.92°, and Reynolds number of 6.5×10^6 based on airfoil chord. The coefficients of lift and drag from the experiment were 0.743 and 0.0127, respectively.

The experimental Cp values are available from the NPARC alliance validation archive webpage https://www.grc.nasa.gov/WWW/wind/valid/raetaf/raetaf04/raetaf04.html. The NPARC simulation was performed with a far-field Mach number of 0.729 and with the angle of attack set to 2.31° to adjust for the influence of the wind tunnel walls. The far-field temperature was set to 460° Rankine and the pressure was 15.80734 PSI. The Reynolds number based on a chord of one foot was 6.5x10^6.

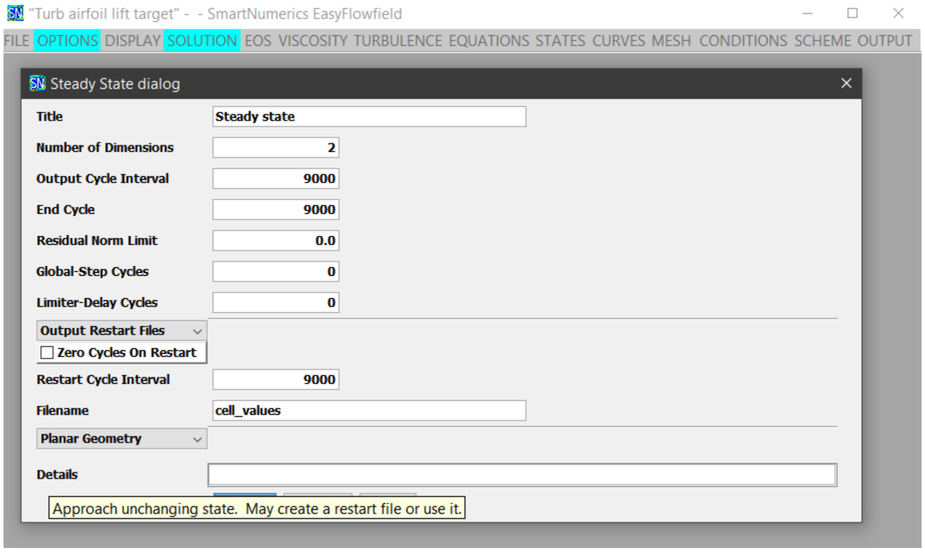

Fig. 1: Specify output interval and end cycle for initial simulation.

First open the Simulation Overview dialog under menu heading **OPTIONS** and change the title to "Turb airfoil lift target". Then open the Steady State dialog under menu heading Solution and set ‘Output Cycle Interval’ and ‘End Cycle’ to 9000. Also select output of restart files with an interval of 9000 cycles.

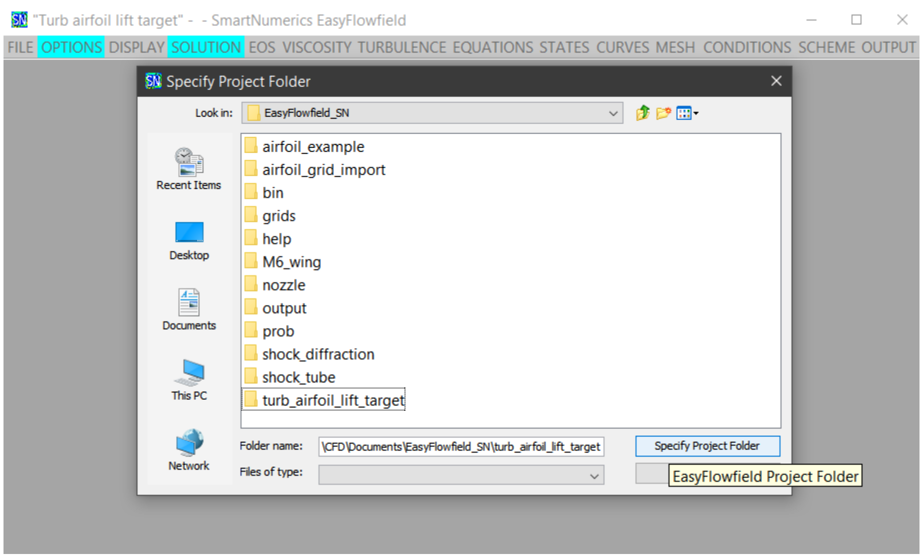

Fig. 2: Create project folder.

Next create a project folder by clicking on Add Project Folder under menu heading **FILE**, navigating to the EasyFlowfield_SN folder, appending the name of the new project folder, and clicking on Specify Project Folder. The new project folder, which in this case is "turb_airfoil_lift_target", will be created as a subfolder of the EasyFlowfield_SN folder and you are prompted to save the new script.

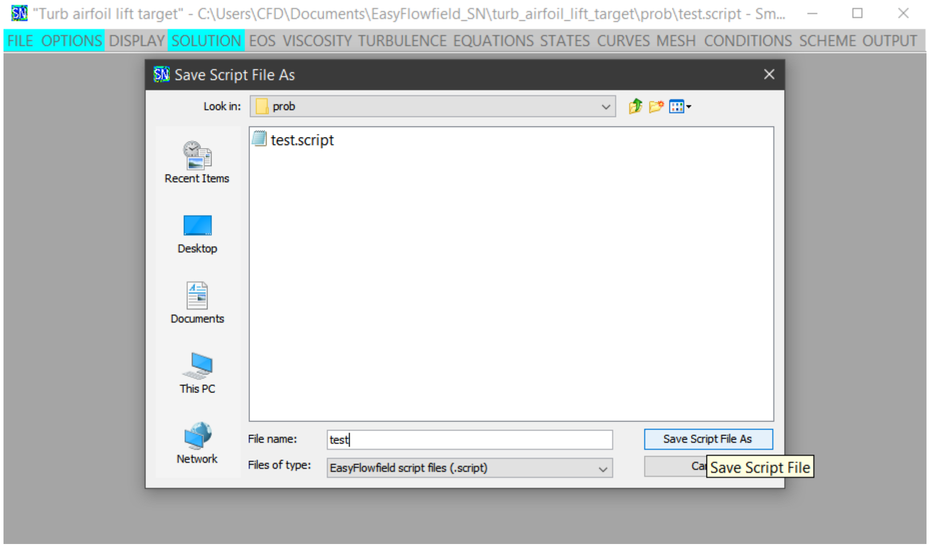

Fig. 3: Save script file.

Note that the turb_airfoil_lift_target folder has the subfolders, grids, output, and prob. To save the script, click on Save Script under menu heading **FILE**, navigate to turb_airfoil_lift_target/prob, enter the filename "test", and click on Save Script File. For this example, you must now place the file RAE2822.SRF in turb_airfoil_lift_target/grids. A copy of this file can be found in EasyFlowfield_SN/grids. Please copy it using Microsoft Windows File Explorer.

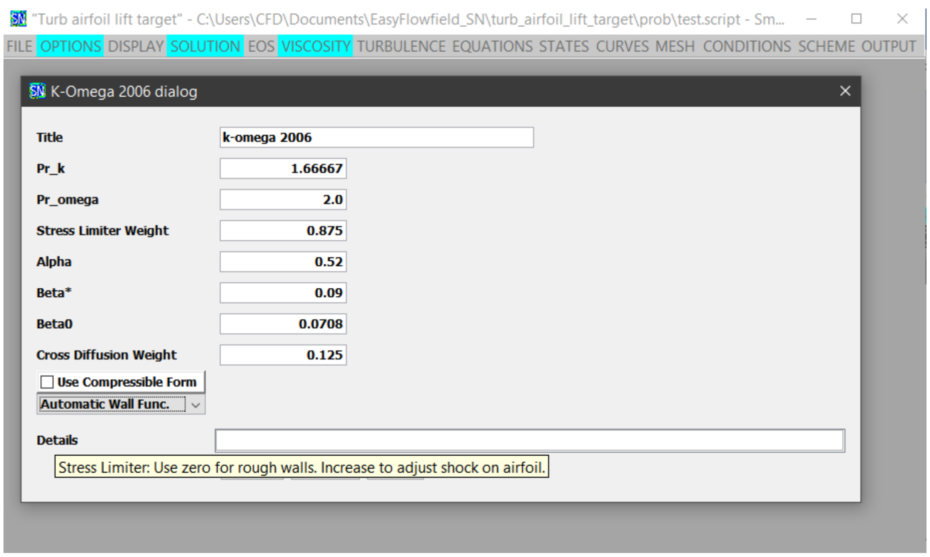

Fig. 4: Specify k-ω 2006 turbulence model with automatic wall function.

Next, please open and close the Cp & Sutherland dialog under menu heading **VISCOSITY** without changing any parameters. Then open the k-omega 2006 dialog under menu heading **TURBULENCE** and specify use of the automatic (adaptive) wall function.

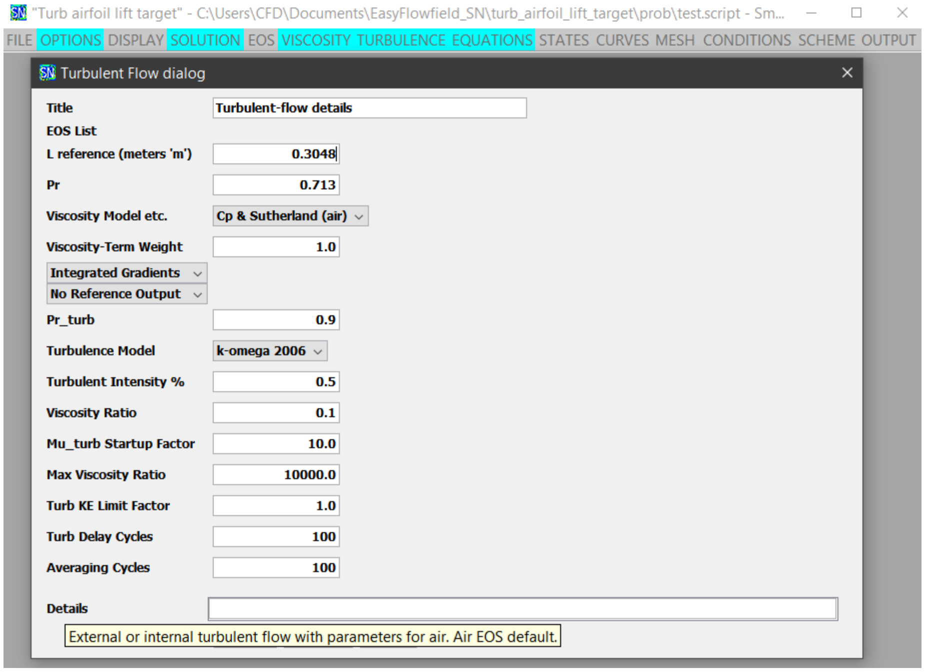

Fig. 5: Select parameters used with simulation of turbulent flow.

Then open the Turbulent Flow dialog available under menu heading **EQUATIONS** and set the reference length to the airfoil chord which will be 0.3048 meters (1 foot). Please close the dialog with all other parameters left at their default values.

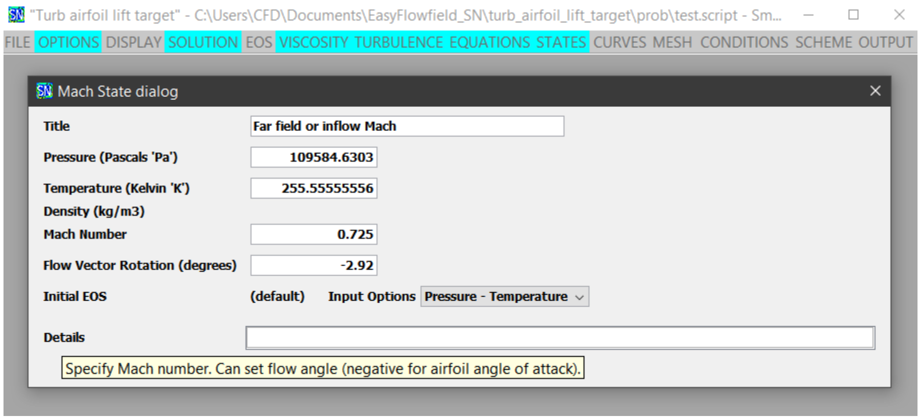

Fig. 6: Specify far-field conditions.

Next open the Mach State dialog available under menu heading **STATE**, set the Mach number to the experimental value of 0.725 and the Flow Vector Rotation to the negative of the experimental value of AOA (2.92). Set the temperature and pressure as indicated in Figure 6. The temperature in degrees Kelvin corresponds to 460 degrees Rankine. The pressure (109584.6303 Pascals) is chosen so that the Reynolds number based on airfoil chord is equal to 6.5x10^6. The solver outputs the Reynolds number at the start of the simulation to simplify trial and error adjustments to the flow parameters.

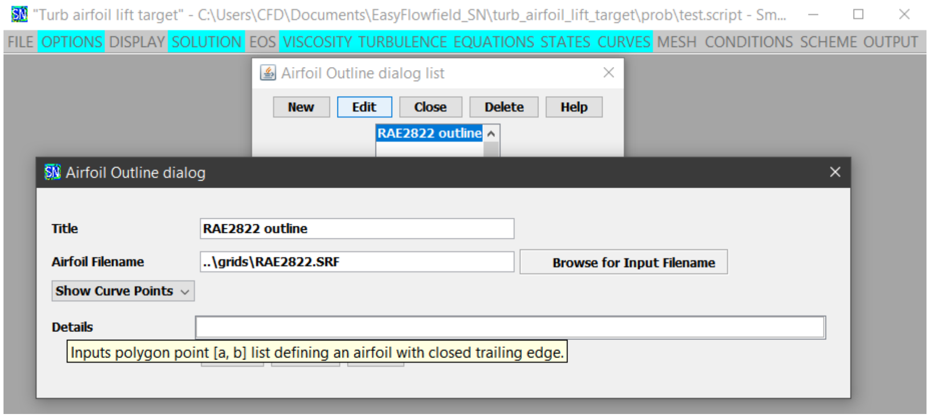

Fig. 7: Input airfoil outline.

Next input a file containing points defining the shape of the airfoil using the Airfoil Outline dialog available under menu heading **CURVES**. In this case, the file conforms to the .srf format. Other file formats can be input as detailed in help under menu heading **CURVES**. The title of the dialog has been changed from the default and display of the outline as a sequence of points has been selected.

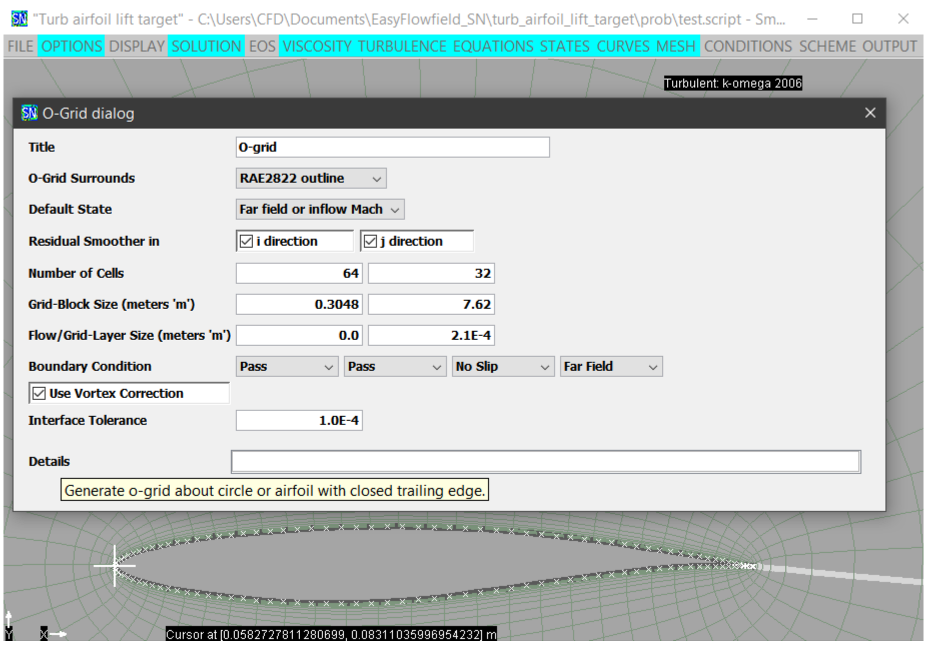

Fig. 8: Creation of o-grid about airfoil.

Next open, the O-Grid dialog under menu heading **MESH,** select "RAE2822 outline" in the drop-down list under the title, and specify a 64x32 cell grid. Please leave all the other parameters at their default values. This will produce a coarse grid. Please set the chord to 0.3048 meters and the far-field boundary distance to 7.62 meters. Set the vertical Flow/Grid Layer size to $2.1x10^{-4}$ meters to concentrate gridlines at the airfoil surface. The boundary condition on the airfoil surface must be reset to No Slip. Please leave all other parameters unchanged. The grid will only appear after you close the dialog. In Figure 8, zoom mode has been used to inspect the correspondence between the grid and the airfoil profile points indicated by ‘x’ marks. This comparison is best done using the ‘Thin Boundary Lines’ option in the Flow Region Display dialog under menu heading **DISPLAY**.

To doublecheck the initial flow values and chord, please save the script and execute it for a few seconds using Automated Simulation under menu heading **FILE**. Stop the simulation by closing the DOS execution window. The Reynolds number based on the reference length of 0.3048 will be displayed near the bottom of the incomplete listing. The value listed should be 6.5000x10^{6}.

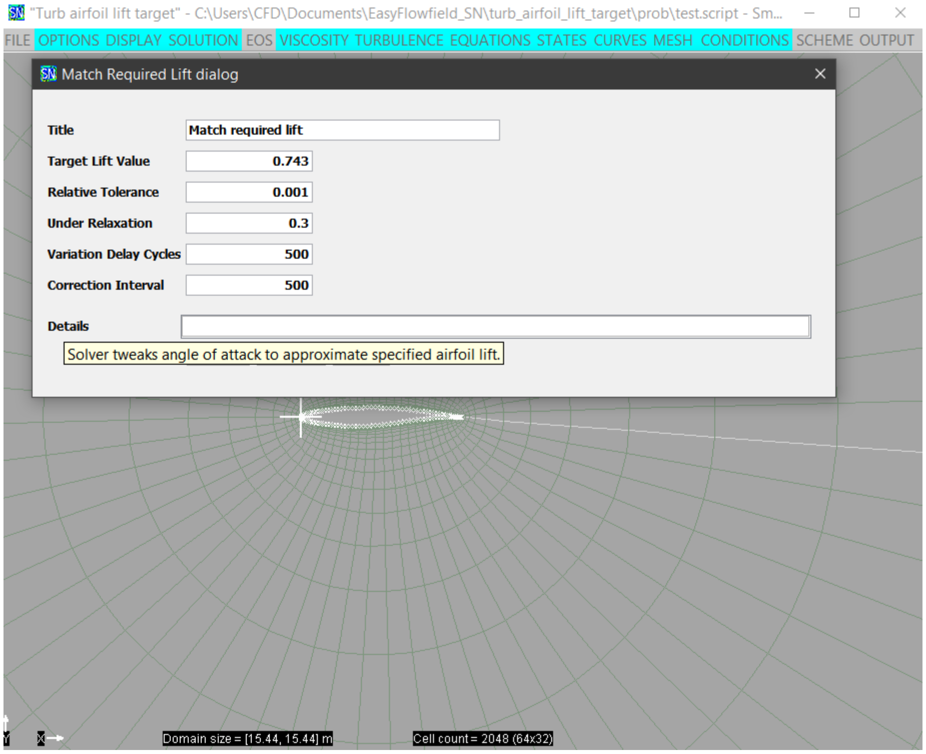

Fig. 9: Specify desired coefficient of lift.

Then open the Match Required Lift dialog under menu heading **CONDITIONS** and set 'Target Lift Value' to the experimental value of 0.473. You can leave the other parameters with their default values. The angle of attack will be tweaked every 500 cycles until the difference between the computed lift and the target lift divided by the target lift is less than the relative tolerance. We want the final lift to be equal to the target lift within plus or minus 0.001. Note that 0.001 divided by 0.743 is approximately 0.0013 which is slightly larger than the specified relative tolerance.

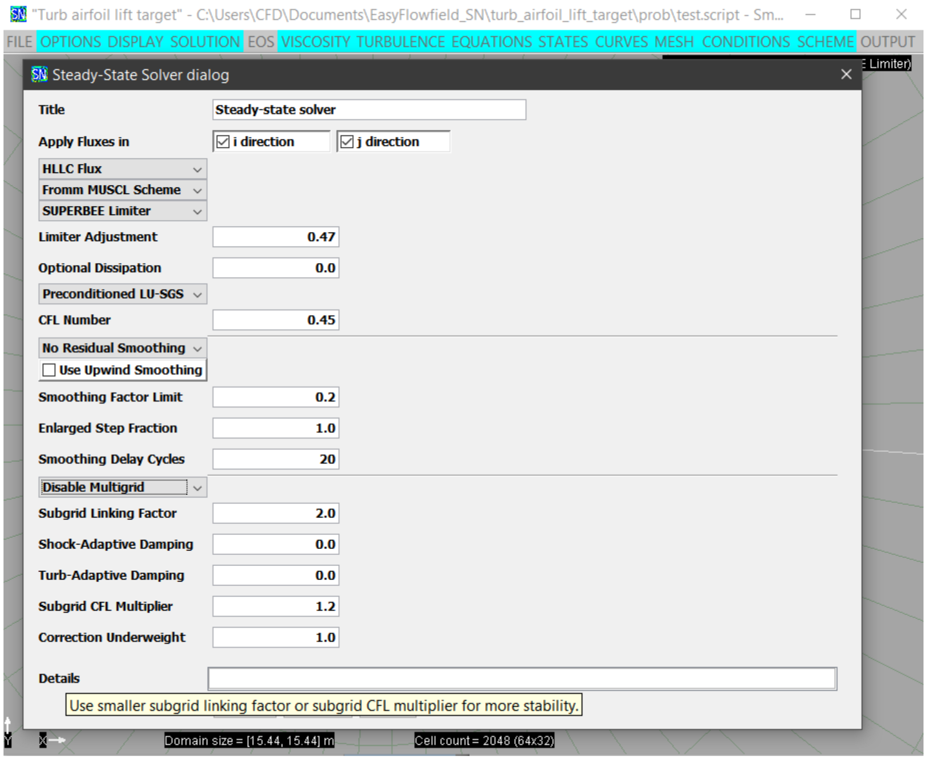

Fig. 10: Specify details of solution scheme.

Next open the Steady-State Solver dialog available under menu heading **SCHEME**, select the SUPERBEE limiter, and set 'Limiter Adjustment' to 0.47. Also select the preconditioned LU-SGS solver and disable multigrid. Please leave all other parameters at their default values.

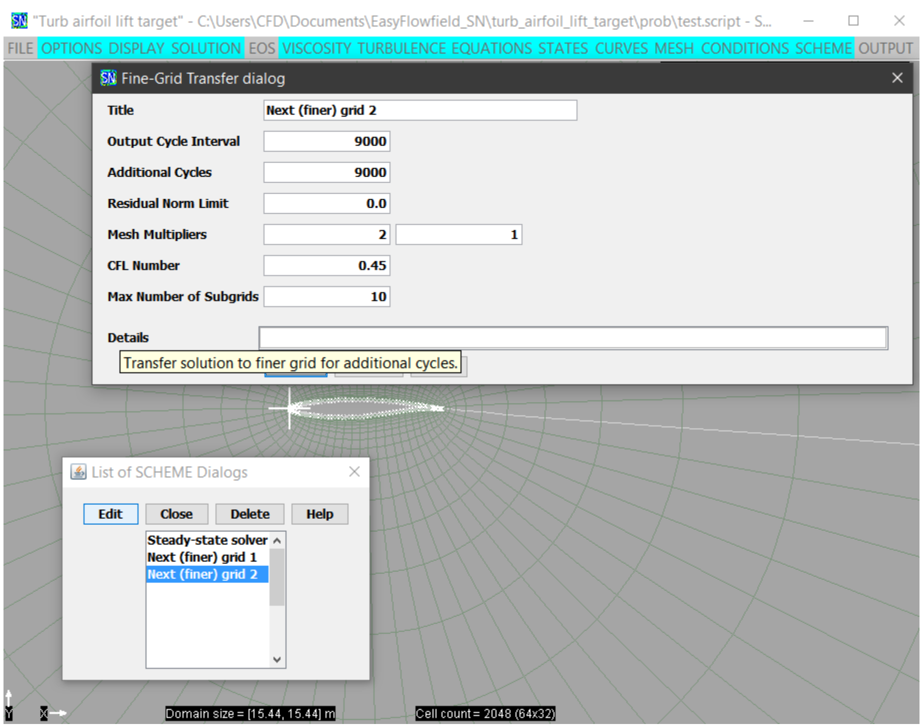

Fig. 11: Specifying simulation on finer grids.

The final solution on the 64x32 cell grid can be used as the initial condition for a simulation on a finer grid. To do this, click on Fine-Grid Transfer under menu heading **SCHEME** and set both 'Output Cycle Interval' and 'Additional Cycles' to 9000. Also set the vertical mesh multiplier to 1. Please leave all other parameters at their default values. In this case, the initial solution will be transferred to a 128x32 cell grid. Please repeat the process using New to specify transfer of the results to an even finer 256x32 cell grid.

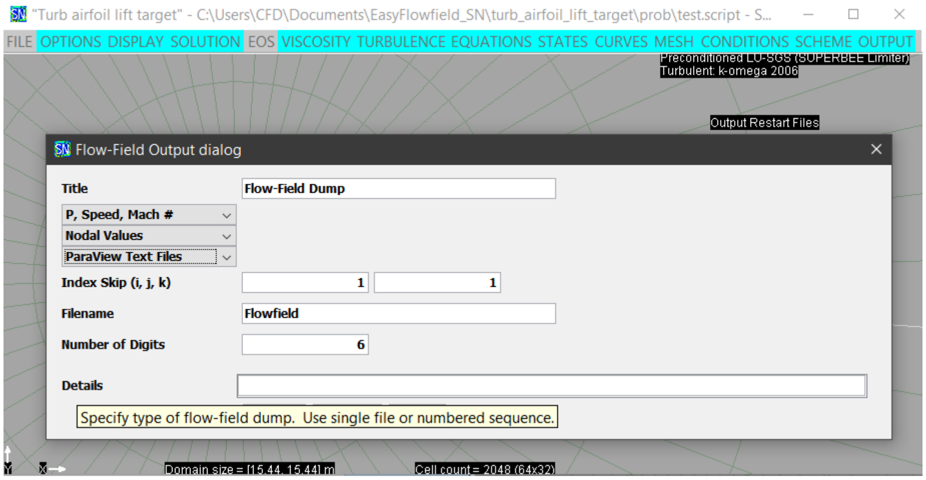

Fig. 12: Specify details of flow-field output.

Then open the Flow-Field Output dialog available under menu heading **OUTPUT**. Specify output of pressure, flow speed, and Mach nuber. Also select output of values at the nodes (cell corners) and use of the ParaView file format.

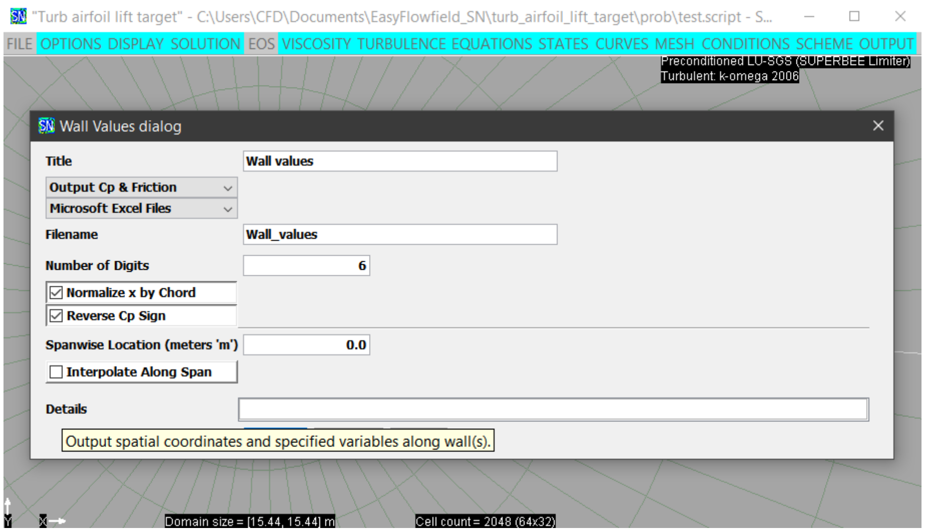

Fig. 13: Specify details for output of values along airfoil surface.

Next open the Wall Values dialog and specify use of the Microsoft Excel file format. Please specify output of pressure coeficients (Cp) and friction coefficients (C_f). Also specify that the coordinates will be normalized by the airfoil chord and that the output values of Cp wll be reversed in sign to match the experimental values.

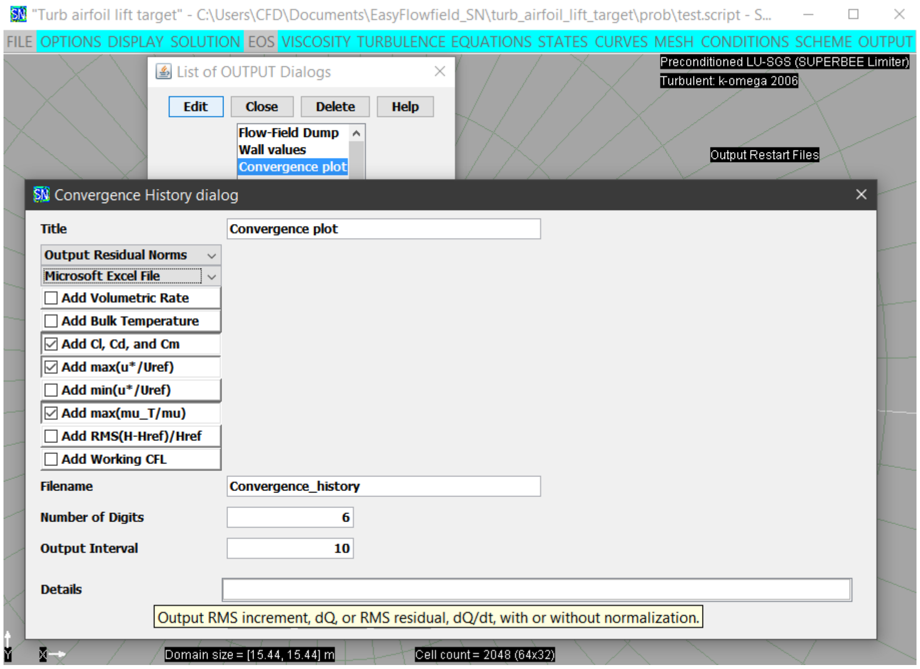

Fig. 14: Specify output used to monitor convergence.

Then open the Convergence History dialog and specify use of the Microsoft Excel file format (.csv). Please select output of the coefficients of lift (Cl), drag (Cd), and pitching moment (Cm). Also add output of the friction velocity (u^*) normalized by the far-field velocity and the eddy viscosity (μ_T) normalized by the molecular viscosity. All other parameters should be left at their default values.

Please close the dialog and save the script. Then run the simulation by clicking on Automated Simulation under menu heading **FILES**. The final value of AOA on the initial, intermediate, and final grids will be respectively 2.25, 2.26, and 2.35 degrees. (2.24, 2.25, 2.34

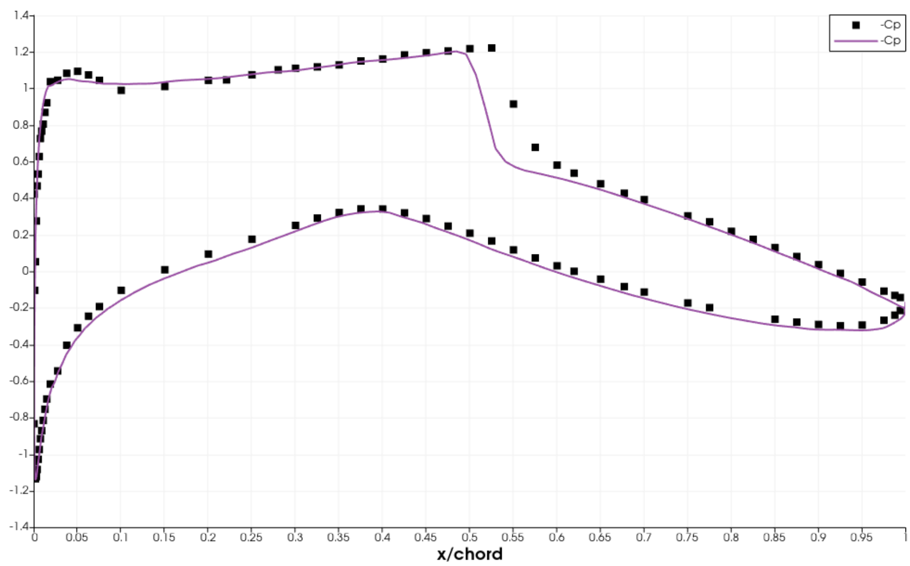

Fig. 15: Cp from finest grid (line) compared to experiment (symbols) (Mach 0.725).

Figure 15 was produced by ParaView by reading a text file containing the experimental values of Cp (squares) and then the wall-values file for the finest grid (wall_values3.csv). A 'Plot Data' filter was applied to each input with 'X Array Name' set to "x/chord" and the dependent variable set to "-Cp". The shock-front location is somewhat upstream of the experimental location. As detailed below, this mismatch will be corrected by increasing the far-field Mach number.

The pressure coefficients from the simulation are computed using

$$c_p = \frac{p_w - p_\infty}{\frac{1}{2}\rho_\infty U_\infty^2}$$

where p_w is the local pressure at the wall, p_∞ is the far-field pressure, ρ_∞ is the far-field density, and U_∞ is the far-field velocity. The experimental data file was created by adding commas and removing comments from a text file originally written in Tecplot format obtained from the NASA website. The filetype of the experimental data files should be set to .txt since EasyFlowfield deletes all files with filetype .dat, .vts, or .csv in the output folder when it initiates a simulation.

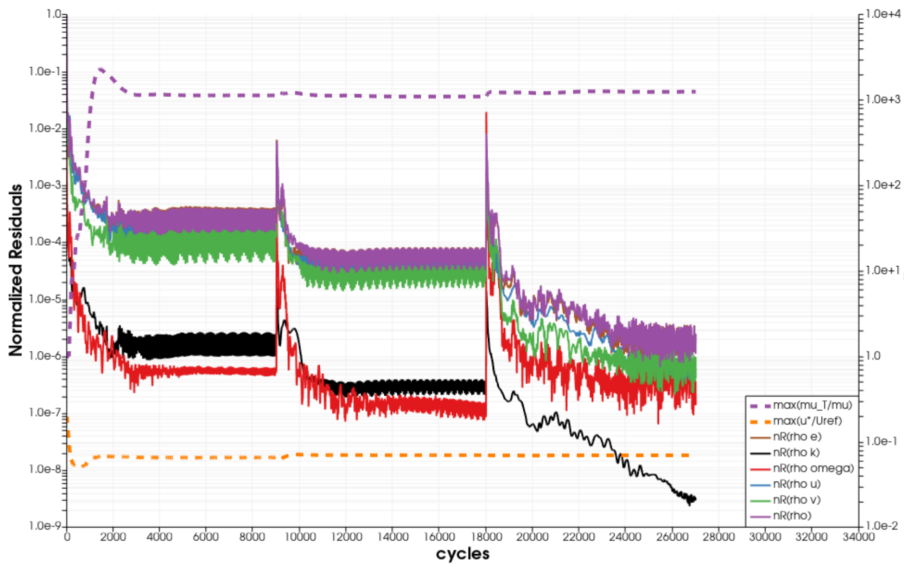

Fig. 16: Convergence history as AOA is adjusted to match lift (Mach 0.725).

Figure 16 displays the convergence history on each grid as AOA is adjusted to match lift. The normalized residuals decrease by more than a factor of 1000 during the simulation on the initial grid and reach smaller values on the subsequent grids. The maximum value of eddy viscosity divided by molecular viscosity is well under the limit of 10,000 specified in the Turbulent Flow dialog and reaches a constant value on each grid. The maximum ration of friction velocity to the far-field velocity also reaches a constant value on each grid.

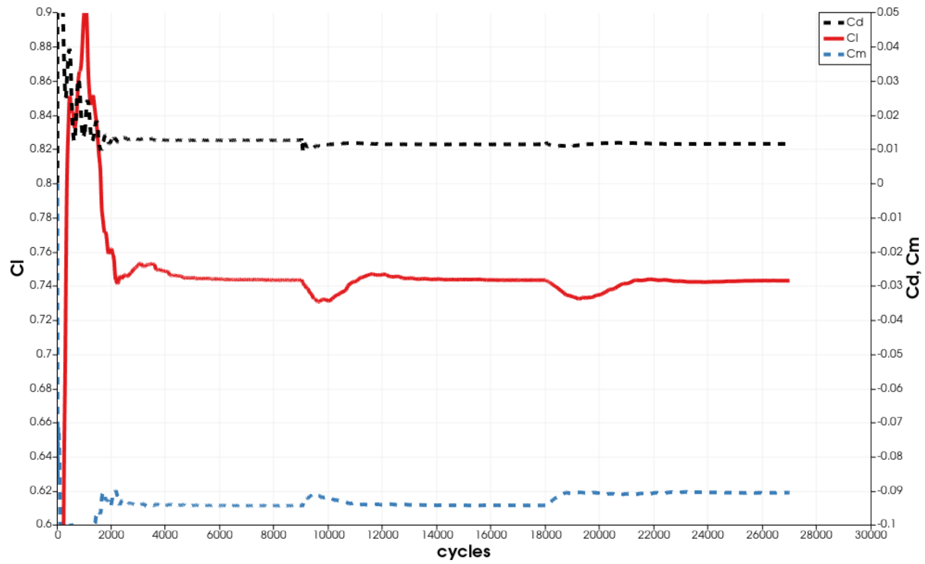

Fig. 17: History of lift and drag as AOA is varied to match lift.

Figure 17 displays the history of lift, drag, and pitching moment as AOA is varied to match lift. Lift varies most rapidly on the initial grid for which the flow angle was set to -2.92 degrees. The flow angle was eventually adjusted to -2.25 degrees on this grid to match the desired value of lift. The lift and drag vary each time the solution is transferred to a finer grid but reach a constant value as the flow angle is adjusted to match the desired value of lift.

Using a process of trial and error, the match between the simulation and experiment can be improved by increasing the far-field Mach number. Please increase the Mach number to 0.729 in the Mach State dialog and run the simulation for a few seconds then close the DOS execution window to halt the simulation. In the listing, the Reynolds number based on the chord of 0.3048 meters will be 6.5359×10^6 which is larger than the experimental value of 6.5×10^6.

The Reynolds number is computed as

$$\mathrm{Re} = \frac{\rho L U}{\mu}$$

where ρ, U, ρ are the far-field values of density, velocity, and molecular viscosity, and L is the reference length. Thus, if velocity increases by certain proportion, the experimental value of Reynolds number can be obtained by reducing density by the same proportion. Since we are using the ideal gas equation of state, this is accomplished by reducing the pressure by the factor 6.5/6.5359. Please enter the resulting value of 108987.8155 Pascals in the Mach State dialog and run the simulation.

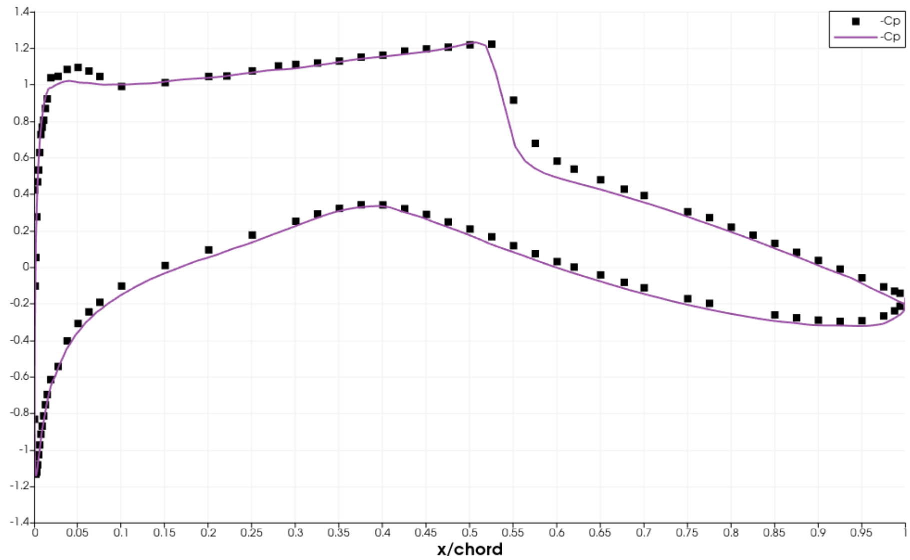

Fig. 18: Cp from finest grid (line) compared to experiment (symbols) (Mach 0.729).

The final value of AOA on the initial, intermediate, and final grids are respectively 2.20, 2.21, and 2.299 degrees. The final value of lift and drag on the finest grid is 0.7434 and 0.0125, respectively.

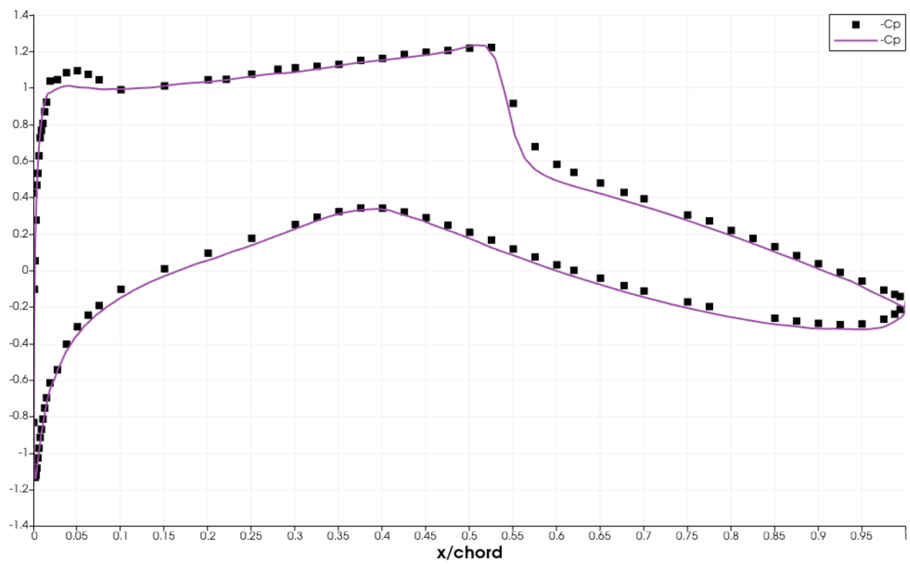

Fig. 19: Cp from finest grid (line) compared to experiment (symbols) (Mach 0.730).

Figure 19 displays the Cp curves with the far-field Mach number increased to 0.730 and the pressure reduced to 108833.7739 Pascals. The final value of AOA on the initial, intermediate, and final grids are respectively 2.18, 2.21, and 2.285 degrees. The final value of lift and drag on the finest grid is 0.7435 and 0.01278, respectively.

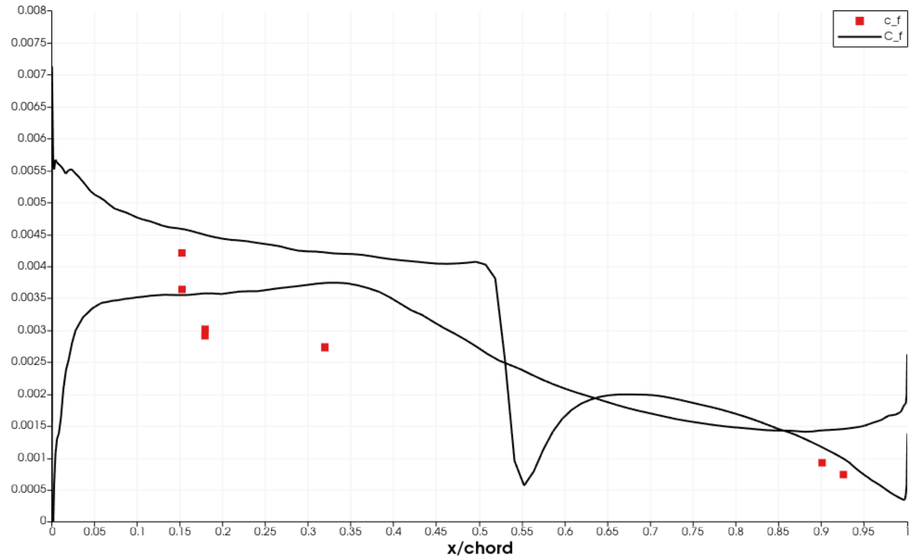

Fig. 20: Provisional C_f from finest grid (line) compared to experiment (symbols) (Mach 0.730).

As seen in Figure 20, the friction is higher than the experimental values taken from Table 6.9 of Cook, McDonald, and Firmin [1]. The friction from the simulation is computed using

$$c_f = \frac{\rho_w (u^*)^2}{\frac{1}{2}\rho_\infty U_\infty^2} sign(u*)$$

where u* is the local friction velocity and ρ_w is the local density at the wall. The friction velocity is given a negative sign if the flow along the wall is reversed. Thus, the plotted friction is negative where the flow reverses. The flow remains attached in this case. There is no reversal of flow seen in the simulation or in the experiment.

According to Holst [2], the experimental C_f values were computed using the dynamic pressure at the edge of the boundary layer instead of the far-field dynamic pressure. The Cp from the simulation should be computed using

$$c_f = \frac{\rho_w (u^*)^2}{\frac{\gamma}{2} p_w M_{is}^2} sign(u*)$$

where the denominator is an approximation to the dynamic pressure at the edge of the boundary layer, γ is the specific heat ratio, and

$$M_{is} = \sqrt{\frac{2}{\gamma-1}\left(\left(\frac{P_{tot}^{\infty}}{p_w}\right)^{(\gamma-1)/\gamma} - 1\right)^{1/2}}$$

is the isentropic Mach number where P_{tot}^{∞} is the freestream total pressure. Unlike the far-field dynamic pressure used in the denominator for *Cp*, the dynamic pressure at the edge of the boundary layer varies along the airfoil.

The ratio of specific heats for air is 1.4 and the total pressure provided in the listing is 155132.77 Pascals. Thus, the dynamic pressure at the edge of the boundary layer is

$$\frac{\gamma}{2} p M_{\text{is}}^2 = p_{\text{w}} 3.5\left(\left(\frac{155132.77}{p_{\text{w}}}\right)^{1/3.5} - 1\right).$$

Friction computed in terms of dynamic pressure at the edge of the boundary layer can be obtained by creating formulas in Microsoft Excel or ParaView to compute the required values.

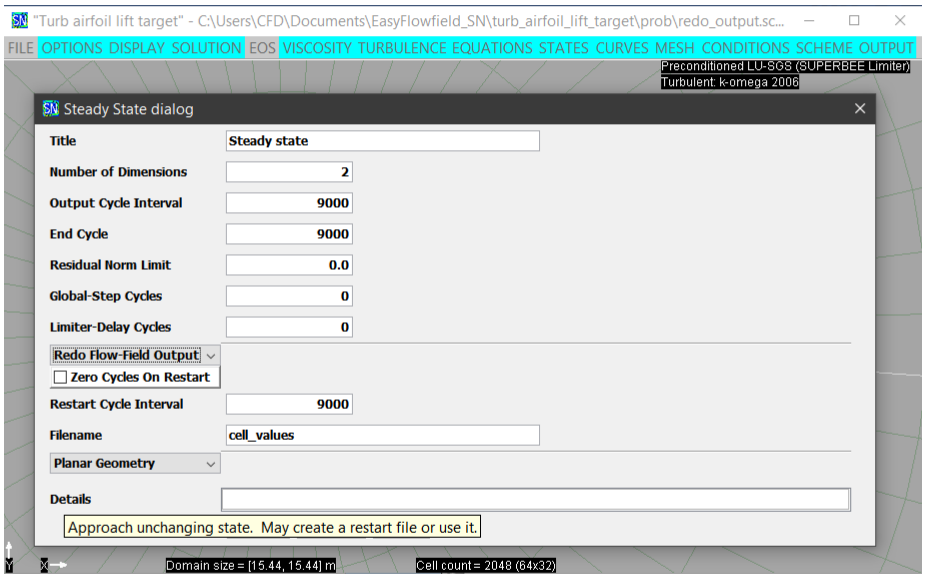

Fig. 21: Selecting 'Redo Flow-Field Output' option.

Save test.script as redo_output.script and select the 'Redo Flow-Field Output' option in the Steady State dialog. This option uses the restart files to repeat output of the flow-field and wall values.

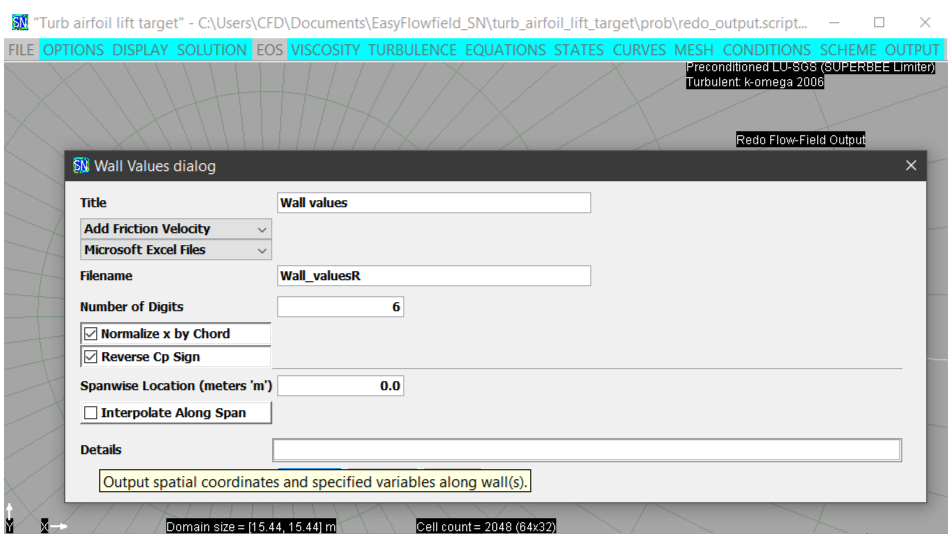

Fig. 22: Change details of wall-values output.

Next, open the Wall Values dialog and specify 'Add Friction Velocity' so that u* will be included in the output. Also change the filename to "Wall_valuesR" to prevent the contents of the Wall_values.vts files from being overwritten. Please close the dialog, save the script, and run the script by clicking on Background Simulation under menu heading **FILES**. Note that the final airfoil forces will not match those originally computed unless you open the Mach State dialog and set the flow angle to -2.28466 degrees.

Start ParaView and open Wall_valuesR_3.vts. Add a calculator filter, specify the 'Result Array Name' "Pdyn", and enter

p (Pa)*3.5*((155132.77/p (Pa))^(1/3.5)-1.0)

as the defining equation. You can copy and paste the above expression. Add a second calculator filter, specify the 'Result Array Name' "C_f", and enter

(rho (kg/m3)*abs(u* (m/s))*u* (m/s))/Pdyn

as the defining equation. Note that the calculator will compute an incorrect value if the numerator is not enclosed by brackets.

To plot the friction values, apply the 'Plot Data' filter. Set 'X Array Name' to "x/chord" and select "C_f" as the dependent variable. Finally, read the text file containing the experimental values of C_f, and apply a second Plot Data Filter with 'X Array Name' set to "x/chord" and "C_f" as the dependent variable.

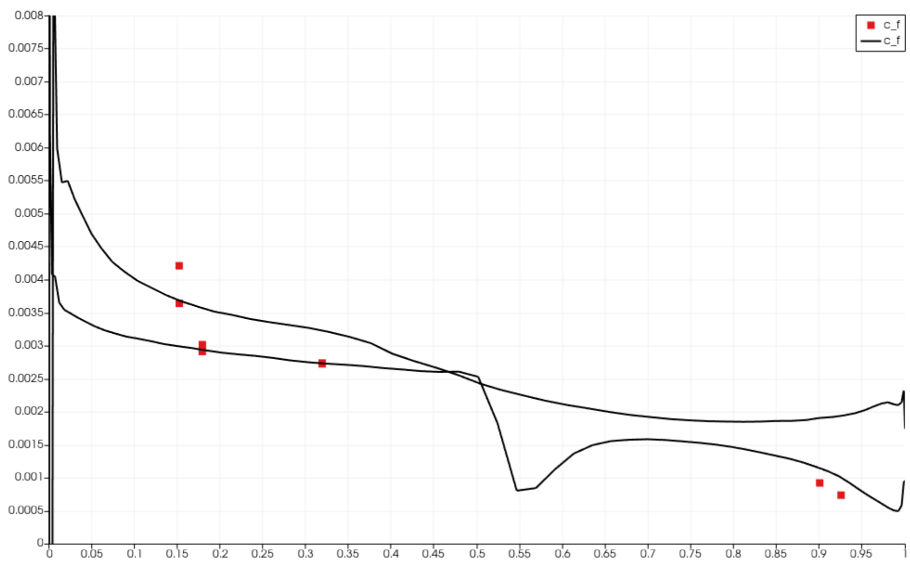

Fig. 23: Corrected C_f from finest grid (line) compared to experiment (symbols) (Mach 0.730).

Figure 23 compares the modified friction curve from the final grid to the experimental values. Normalizing by the dynamic pressure at the boundary-layer edge rather than by the far-field dynamic pressure, has resulted in a much better match to experiment.

A less satisfactory way of plotting C_f, is to use flow-field output. First, open the Flow-Field Output dialog, specify 'Add Gradients or y+' and select output of cell values so that u* will be included in the output. Also change the filename to "FlowfieldR" to prevent the contents of the Flowfield.vts files from being overwritten. Please close the dialog, save the script, and run the simulation by clicking on Background Simulation.

Start ParaView and open FlowfieldR_3.vts. Apply the filter 'Extract Subset' and set the upper j index to 0. This extracts the cells next to the airfoil surface.

Add a calculator filter, specify the Result Array Name 'x/chord", and enter coordsX/0.3048.

as the defining equation. Next, add a second calculator filter, specify the Result Array Name "Pdyn", and enter

p (Pa)*3.5*((155132.77/p (Pa))^(1/3.5)-1.0)

as the defining equation. Finally, add a third calculator filter, specify the Result Array Name "C_f", and enter

(rho (kg/m3)*abs(u* (m/s))*u* (m/s))/Pdyn

as the defining equation.

To plot the friction values, apply the 'Plot Data' filter. Set 'X Array Name' to "x/chord" and select "C_f" as the dependent variable. To compare the results to experimental data use 'Save Data' to export wall values as a .csv file. Please open another instance of ParaView, read the text file containing the experimental values of C_f, and then the new .csv file. Create a comparison plot as previously.

References

[1] Cook, P. H., McDonald, M. A., Firmin, M. C. P., "Aerofoil RAE 2822 - Pressure Distributions, and Boundary Layer and Wake Measurements," in Experimental Data Base for Computer Program Assessment, AGARD Report AR 138, 1979.

[2] Holst, T. L., Viscous Transonic Airfoil Workshop Compendium of Results, AIAA-87-1460, 1987.