EasyFlowfield Tutorial 1: Simulating Steady-State Inviscid Flow About an Airfoil

SmartNumerics Simulation Solutions Inc.

Version 6

June 4, 2020

Copyright SmartNumerics Simulation Solutions Incorporated © 2020, All Rights Reserved.

In this exercise you will simulate inviscid flow about a NACA 0012 airfoil. You will obtain values of lift and drag along with a plot of surface pressure coefficients. The basics of performing a steady-state simulation will be presented along with application of multigrid to accelerate convergence. Use of a succession of finer grids illustrates the concept of grid conference. Simulation of laminar and turbulent flow is also discussed.

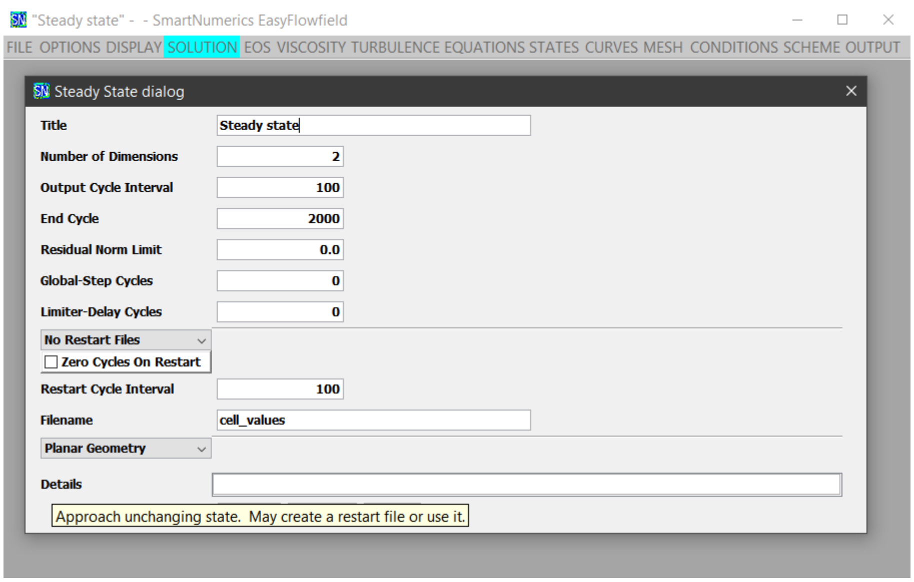

Fig. 1: Specifying type of simulation and number of dimensions.

First specify a two-dimensional steady-state simulation by opening and closing the Steady State dialog under menu heading **SOLUTION**. Please leave all parameters at their default values. Then open the Simulation Overview dialog under menu heading **OPTIONS,** change the title to "Airfoil Example", and close the dialog. This will affect the information displayed along the top of the EasyFlowfield GUI.

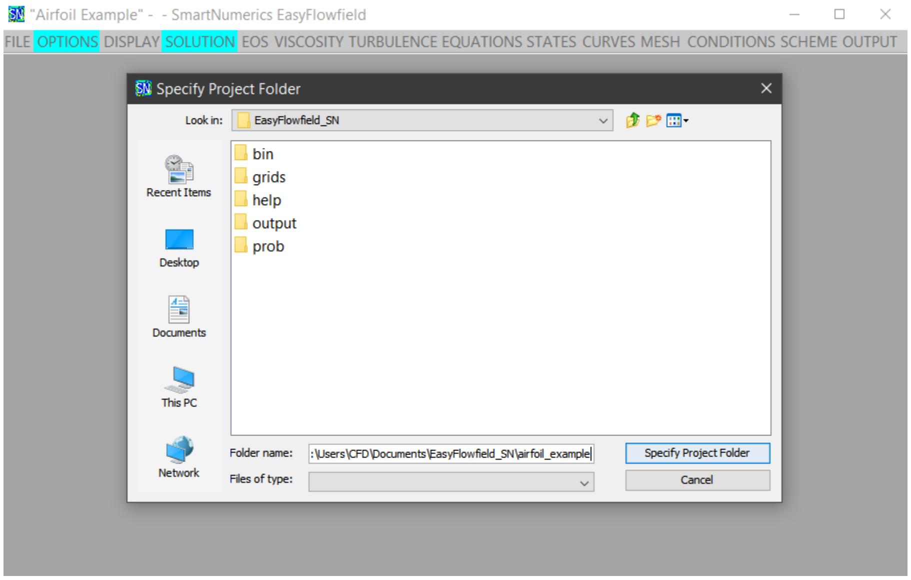

Fig. 2: Create project Folder.

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 "airfoil_example", will be created as a subfolder of the EasyFlowfield_SN folder and you will be prompted to save the new script.

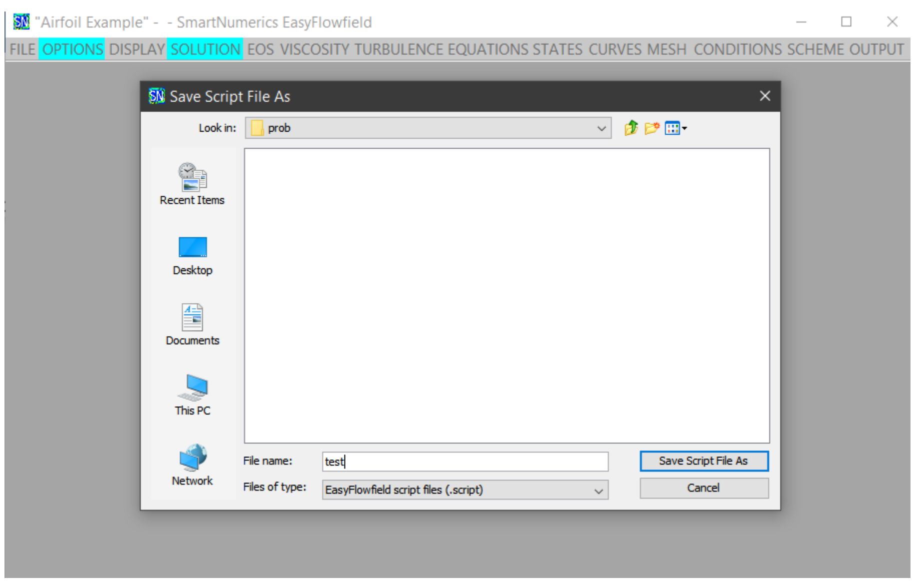

Fig. 3: Saving new script file.

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

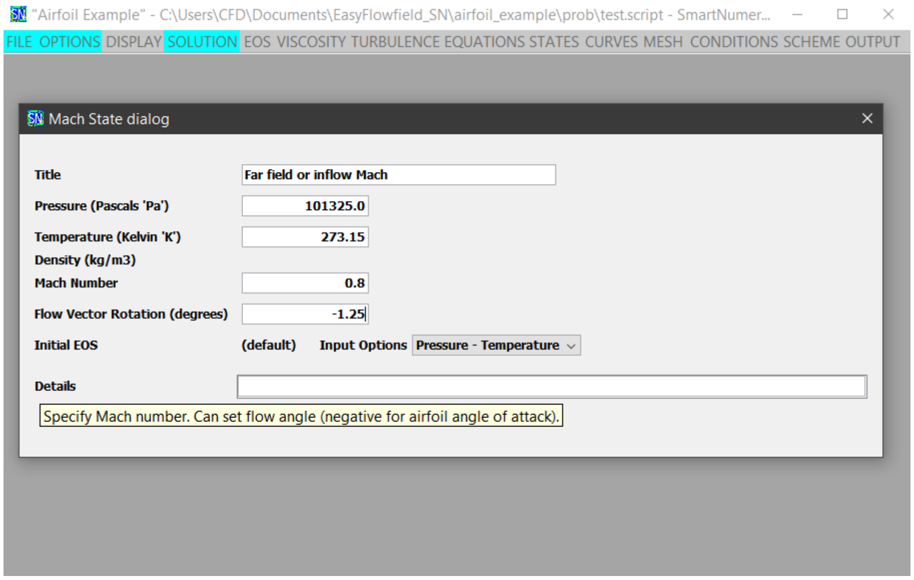

Fig. 4: Specifying far-field conditions.

Next create a Mach State dialog under menu heading **STATES** and specify a Mach number of 0.8 and a flow angle of -1.25 degrees, leaving pressure and temperature at their default values.

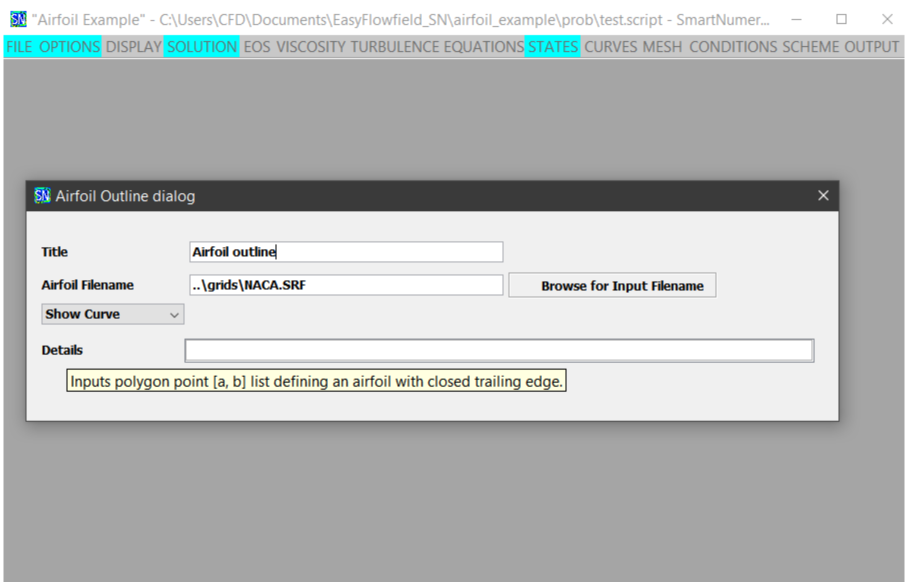

Fig. 5: Input airfoil outline or profile.

Next input the 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 its default name. The non-default display option 'Show Curve Points' has been selected.

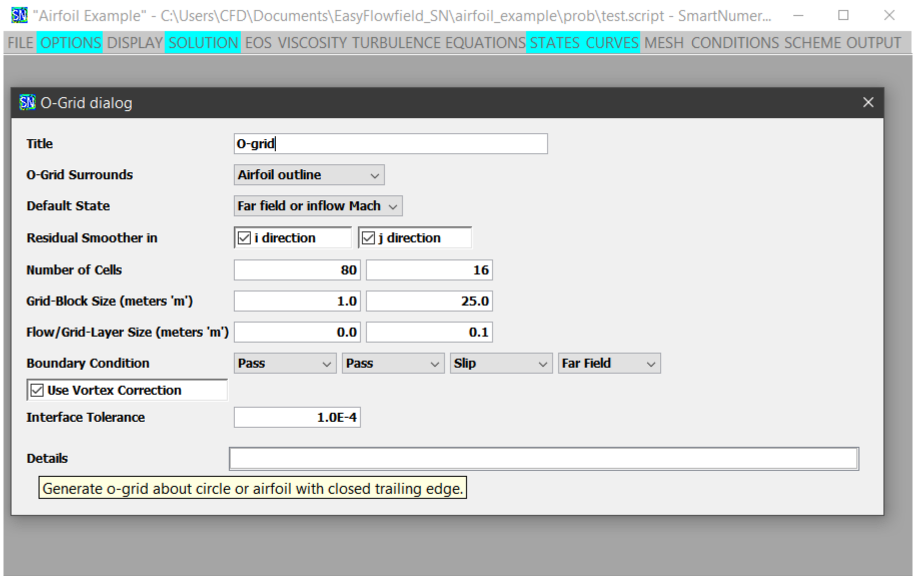

Fig. 6: Creation of o-grid about airfoil.

Next open, the O-Grid dialog under menu heading **MESH** and select "NACA 0012 airfoil outline" in the drop-down list under the title. Please leave all the other parameters at their default values. This will produce a coarse 80x16 cell grid. As demonstrated below, a more accurate solution can be obtained by doubling or quadrupling the number of cells in each direction. The grid will only appear after you close the dialog.

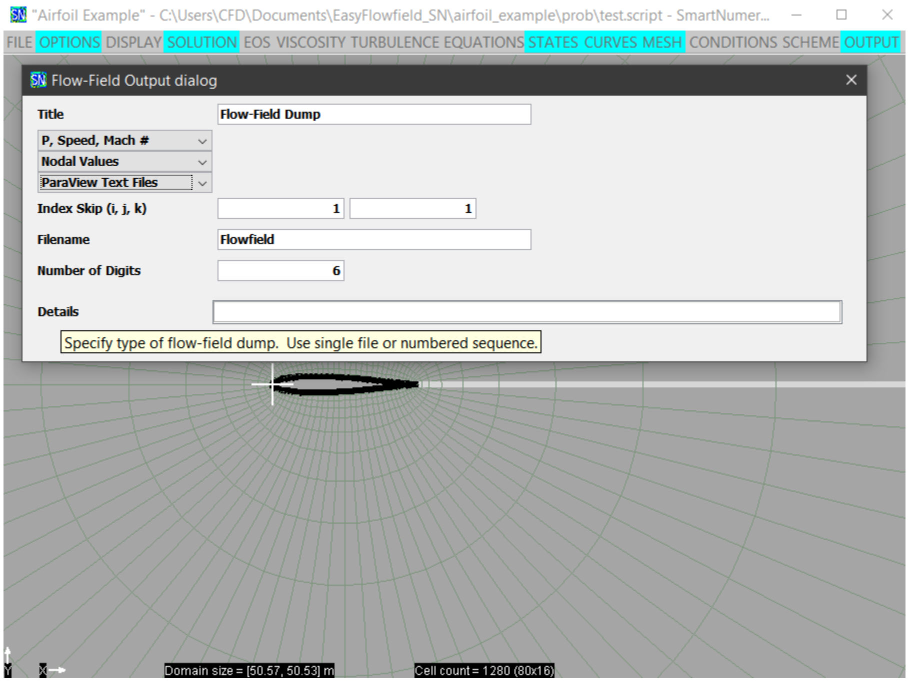

Fig. 7: Specify output for contour plotting.

Next open the Flow-Field Output dialog under menu heading **OUTPUT** and select the 'P, Speed, Mach #' option to output values of pressure, flow speed, and Mach number for contour plotting. Also select 'Nodal Values' (cell corners) and the ParaView text file format option.

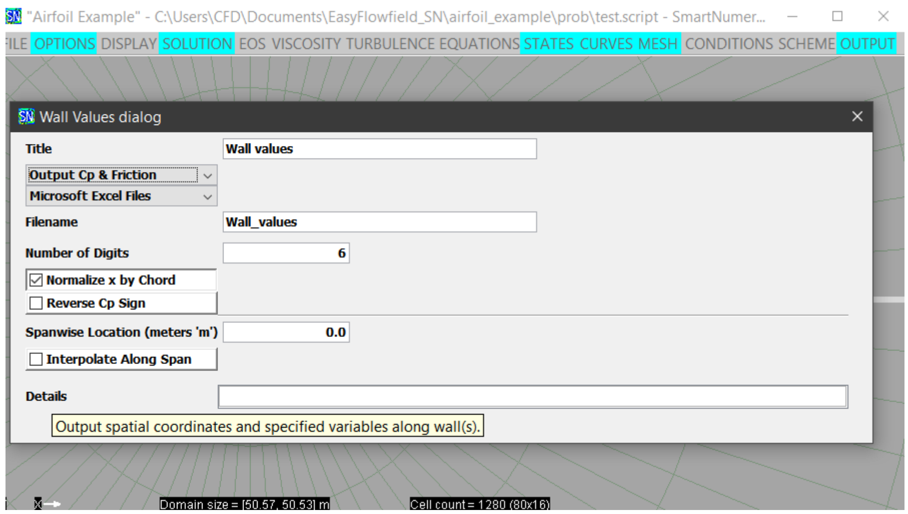

Fig. 7: Specify output for plotting pressure coefficient on airfoil surface.

Next open the Wall Values dialog also under menu heading **OUTPUT,** select the 'Output Cp & Friction' option, and activate the 'Normalize x by Chord' checkbox. This will produce values of Cp versus x divided by the airfoil chord (1 meter) along the airfoil surface. Also, please select the Microsoft Excel file output format. Values will be output in a text file using the "comma separated values" (.csv) format.

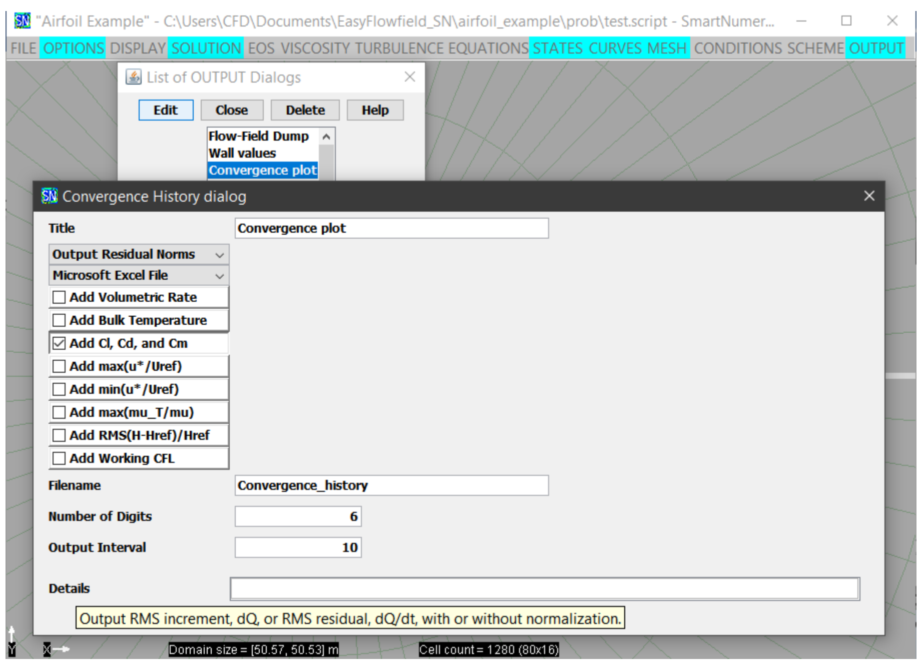

Fig. 8: Specify output used to monitor convergence.

Next open the Convergence History dialog and select output of the airfoil force coefficients Cl, Cd, and Cm. Please also select the Microsoft Excel output format.

Finally click on Save under menu heading **FILE**. The **FILE** menu heading will turn green to indicate that the script has been saved. It will turn grey if you create a new dialog or open an existing dialog to examine the parameter values.

Please run the simulation using Interactive Simulation or Automated Simulation under menu heading **FILE**. In either case, the output files will be in airfoil_example/output. You can open the .vts files using ParaView and the .csv files using Microsoft Excel or ParaView.

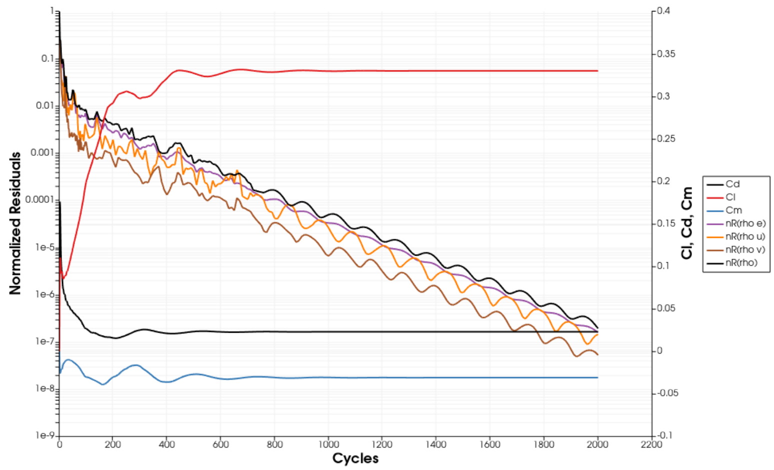

Fig. 9: Convergence history using five-stage Runge-Kutta solver on 80x32 cell grid.

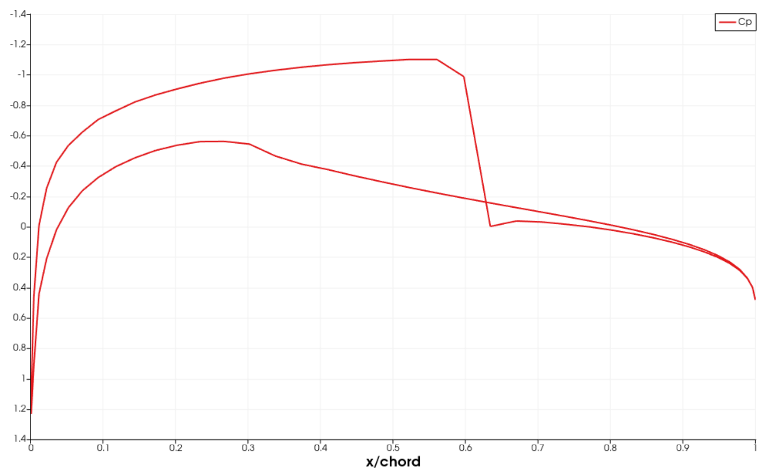

Fig. 10: Cp along airfoil surface using five-stage Runge-Kutta solver on 80x32 cell grid.

Figures 9 and 10, respectively, display a convergence history plot and a pressure coefficient plot produced using ParaView. The Cl, Cd, and Cm profiles were assigned to the right axis by highlighting them in the ParaView list of variables and specifying 'Bottom-Right' instead of 'Bottom-Left'. The coefficients of lift (Cl), drag (Cd), and pitching moment (Cm) are also output in the listing produced by the solver. Figure 11 displays a contour plot of Mach number produced by ParaView. Note how subsonic flow accelerates to supersonic flow on top of the airfoil. The region of supersonic flow is terminated by a shock front.

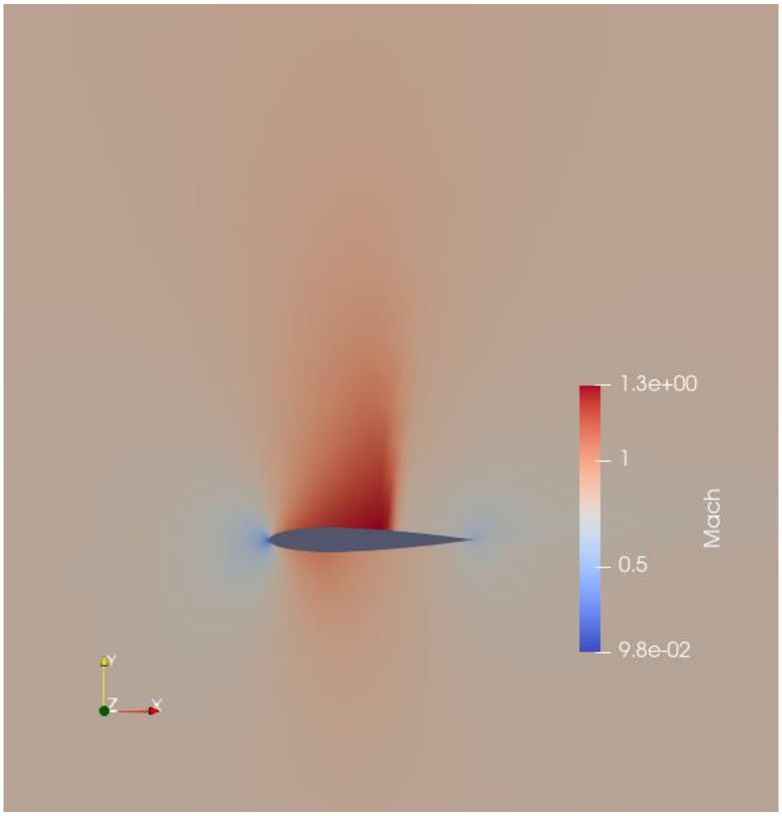

Fig. 11: Mach contours on 80x32 cell grid produced by ParaView.

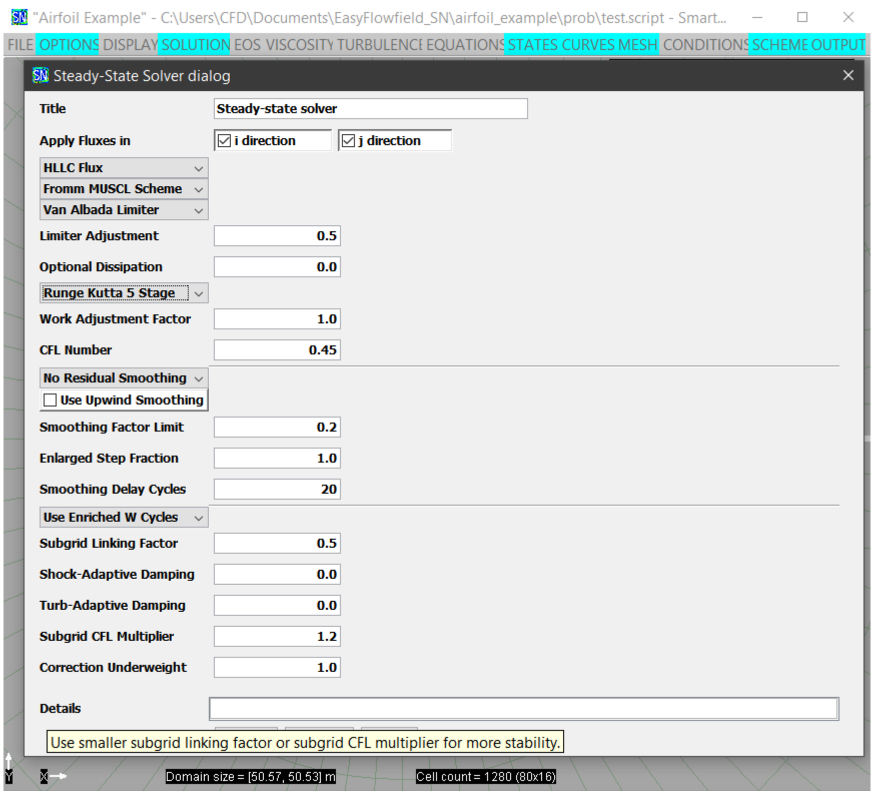

Fig. 12: Dialog controlling details of steady-state simulation.

Faster convergence can be obtained by using a multigrid scheme. First open the Steady-State Solver dialog under menu heading **SCHEME**, select ‘Use Enriched W Cycles’, and reduce ‘Subgrid Linking Factor’ to 0.5 to avoid instability. Please leave all the other parameters at their default values.

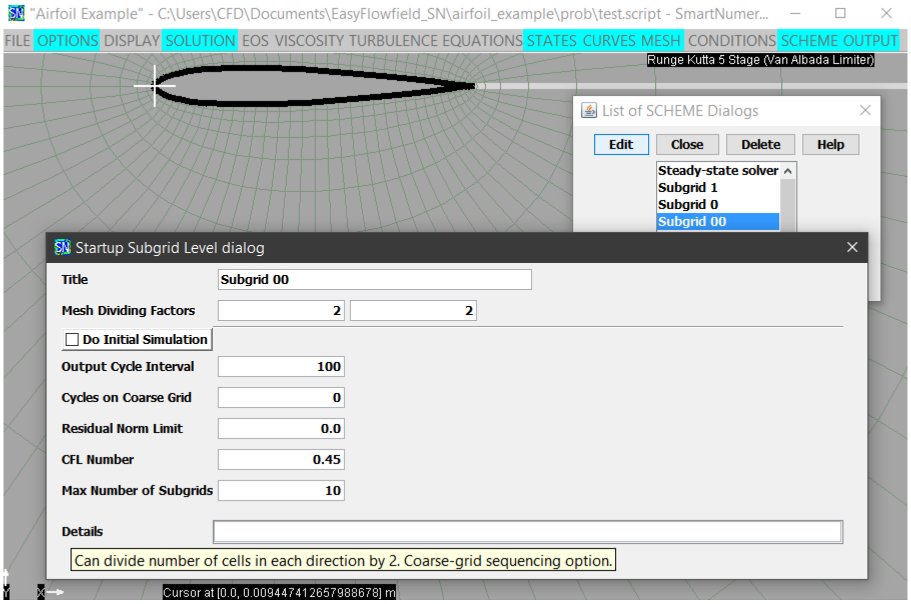

Fig. 13: Creation of coarsest subgrid for multigrid scheme.

Then click on Startup Subgrid Level to create a subgrid with half the number of cells in each direction. Please leave all parameters at their default values. Then close the dialog and use New to create an even coarser subgrid with a slightly different name. Please do this one more time to duplicate the Figure 13 above. Then save the script and rerun the simulation.

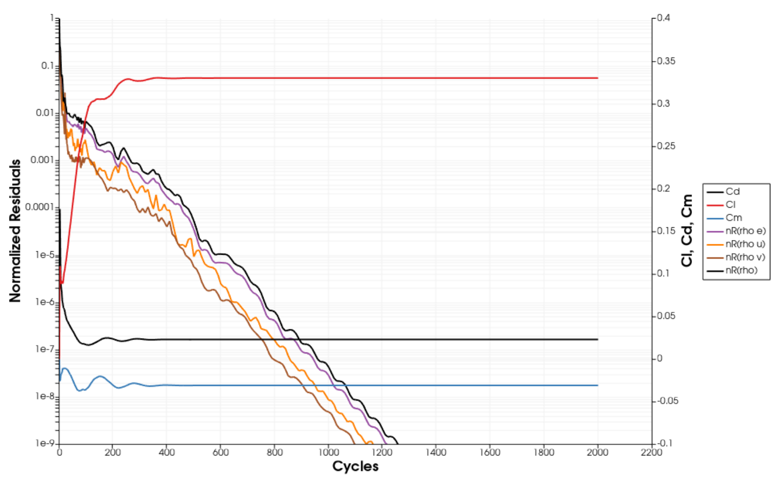

Fig. 14: Convergence history using five-stage Runge-Kutta solver with multigrid on 80x32 cell grid.

Figure 14 displays the convergence history with multigrid acceleration to the steady state. The lift, drag, and moment force coefficients have reached constant values well before 1000 cycles.

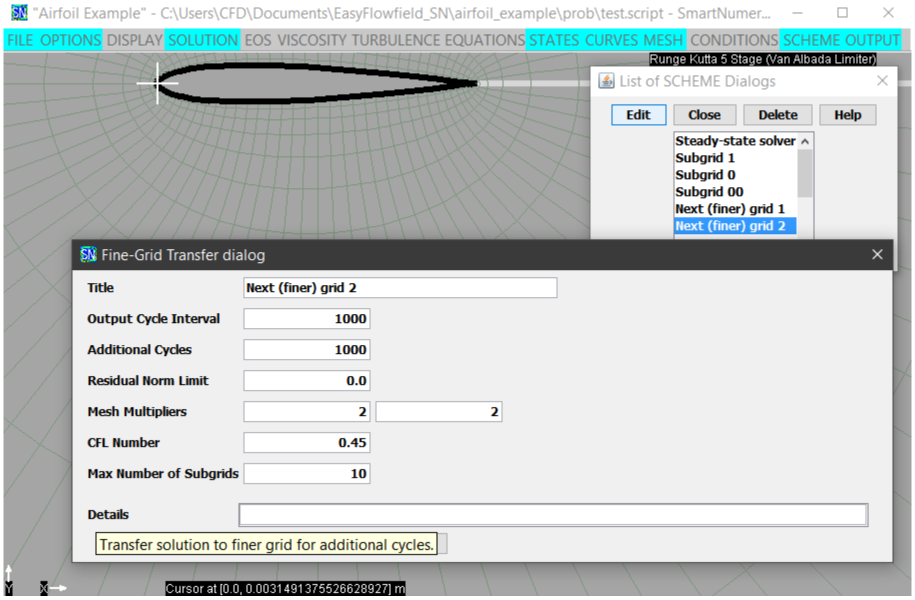

Fig. 15: Specifying simulation on finer grids.

The final solution on the 80x32 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 1000. Please leave all other parameters at their default values. In this case, the solution will be transferred to a 160x64 cell grid. Repeat the process to specify transfer of the results to an even finer 320x128 cell grid. Please reopen the Steady State dialog under menu heading **SOLUTION** and reduce 'End Cycle' to 1000. Please save the script and rerun the simulation.

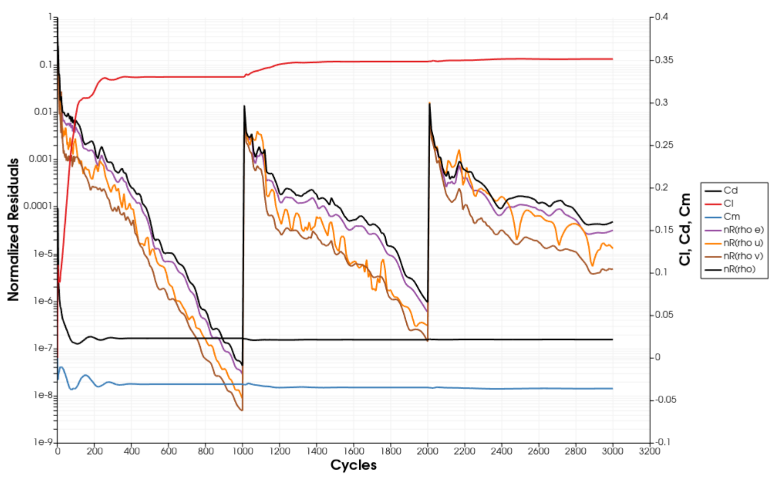

Fig. 16a: Convergence using five-stage Runge-Kutta solver, multigrid, and two finer grids.

Figure 16a displays the resulting convergence history. Performing a simulation on a sequence of finer grid is typically called a "grid convergence study" or "grid refinement study". As the number of cells is increased, the spatial and temporal discretization errors should become smaller. At some point, use of a finer grid will not significantly improve the accuracy of the solution. In this case, use of an even finer grid will only have a minor effect on the coefficient of lift.

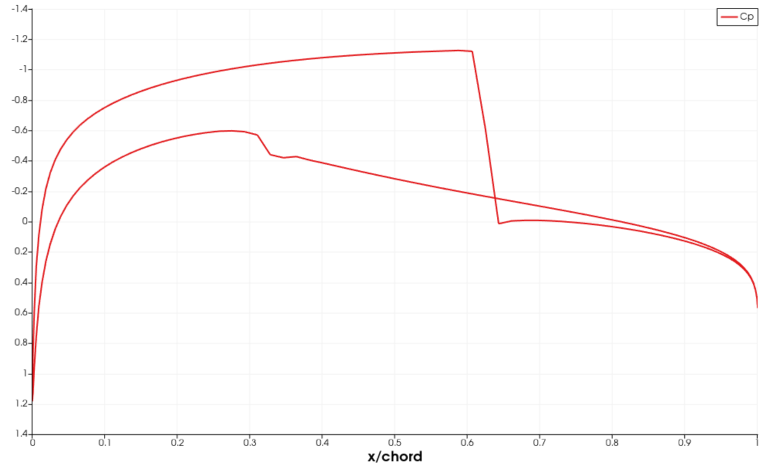

Fig. 16b: Cp along airfoil surface using five-stage Runge-Kutta solver on 160x64 cell grid.

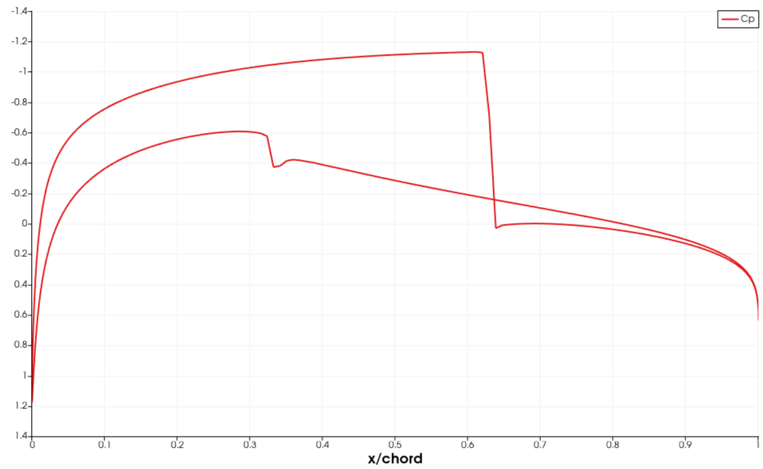

Fig. 16c: Cp along airfoil surface using five-stage Runge-Kutta solver on 320x128 cell grid.

Figures 16b and 16c display the pressure coefficients on the second and third grids. Note how the thickness of the shock front decreases as the number of cells along the airfoil surface is increased. Figure 16d displays the Mach contours on the final grid.

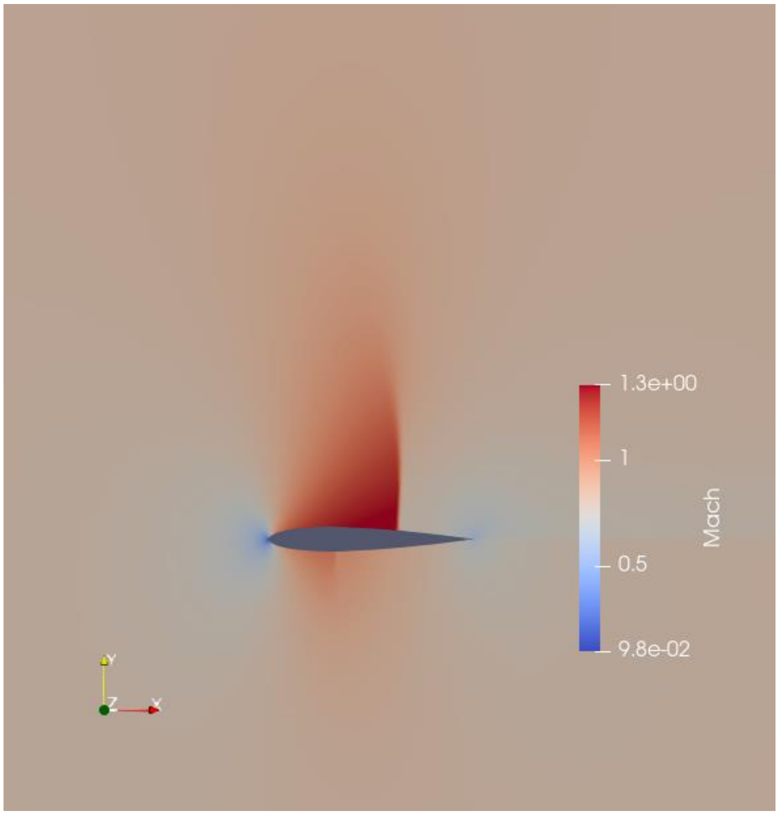

Fig. 16d: Mach number contours on finest grid produced by ParaView.

At this point it is useful to save a copy of the current script file. Please click on Save Script As under menu heading **FILE** and save the script using a different name such as test_explicit.script. Please then reopen test.script by clicking on Read Script. The script test_explicit.script can be run at a future date by opening it and clicking on Background Simulation under menu heading **FILE**.

To use an alternative solver, please reopen the Steady-State Solver dialog, replace 'Runge-Kutta 5 Stage' by 'Preconditioned LU-SGS' and set the 'Subgrid Linking Factor' to its original value of 2. Then open the Implicit-Step Options dialog under menu heading **OPTIONS**, set 'Max CFL Enhancement' to 100, and 'Enhancement Delay' to 100 cycles. Please save the script and rerun the simulation.

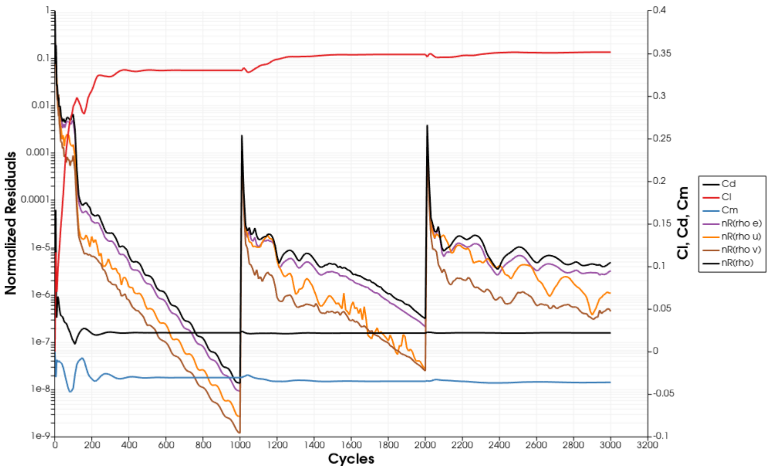

Fig. 17: Convergence history using preconditioned LU-SGS, multigrid, and finer grids.

Figure 17 displays the convergence history using the preconditioned LU-SGS solver. Note the rapid decrease in the residual values near 100 cycles when the CFL number increases from 0.45 to 45.

You should now try using a slightly smaller value of 'Limiter Adjustment' such as 0.45 using the Steady-State Solver dialog to see the effect on the thickness of the shock front and on the lift and drag coefficients. Before you make this change, please click on Save Script As under menu heading **FILE** and save the script using the name test_implicit.script then reopen test.script. You should also try using different limiters or the first order upwind flux.

EasyFlowfield can simulate laminar and turbulent flow. Some sample scrips have been provided in the folder EayflowField_SN/prob in read-only form. You can open and run the scripts using the EasyFlowfield GUI. Output will appear in EasyflowField_SN/output. If you wish to make a change in a sample script, please use “Add Project Folder” to create a project with a matching name (e.g., house) in the EasyFlowfield_SN folder. Then use “Save As” to save the sample script as test.script in the prob subfolder of the new project (e.g., house/prob). If the sample script requires input of a curve or outline it should first be copied from EasyFlowfield_SN/grid to the grid subfolder of the new project using Microsoft Windows File Explorer. Some of the scripts contain explanatory comments. They usually appear in a solver dialog under menu heading **SCHEME**. Any additional comments are most easily found by running the script using the “Background Simulation” option and examining the first part of the listing.

The file airfoil_laminar_Mach_0005.scrip is used to simulate Mach 0.0005 laminar flow about a NACA 0012 airfoil. Note the presence of dialogs under menu headings **VISCOSITY** and **EQUATIONS** that handle details of the laminar flow. This test case uses a higher clustering of gridlines near the airfoil surface to capture the physics of flow in the laminar boundary layer.

The file turb_airfoil_RAE2822.scrip is used to simulate turbulent transonic flow about the RAE 2822 airfoil. Note the presence of dialogs under menu headings **VISCOSITY**, **TURBULENCE**, and **EQUATIONS** that handle details of the turbulent flow. This test case uses a much higher clustering of gridlines near the airfoil surface to capture the physics of flow in the boundary layer.

The file turb_airfoil_AOA.script controls the variation of angle of attack (AOA) from zero to 24 degrees. The output can be used to produce a plot of lift and drag versus AOA or plot of drag versus lift. This test case uses a wall function to reduce the need to cluster grid lines near the airfoil surface.

Note that conditions for laminar and turbulent flows about airfoils are usually expressed in terms of angle of attack, Mach number, and the Reynolds number based on the airfoil chord. For laminar flow, different values of Mach number will produce the similar lift and drag coefficients as long as the chord is adjusted to obtain the same Reynolds number. The Mach number can be larger than the experimental Mach number as long as compressibility effects are negligible. In general, the compressibility effects are minor if the Mach number is less than 0.3. However, values of Mach number and airfoil chord close to experimental or actual flight values should be used when simulating turbulent flows. Please see the list of validation documents available under menu heading **FILE** for more information on turbulent flow about airfoils.