EasyFlowfield Tutorial 4: Flow about M6 Wing
SmartNumerics Simulation Solutions Inc.

Version 6
September 18, 2021

Table of Contents

This tutorial details the imposition of boundary conditions and block connections to a fairly complex imported 3D multiblock grid. The export of subgrids in PLOT3D and CGNS format is also covered. Details related to simulating flow about a wing are covered. Please look at tutorials 1 and 2 first. This is a rather elaborate tutorial that can be skipped unless you are particularly interested in simulating flow over a wing. **Please read the validation document on turbulent flow about the M6 wing first.**

In this lesson you will learn how to import a multiblock grid stored in PLOT3D format and impose appropriate boundary conditions and block interfaces. A grid file must be downloaded from the web site https://www.grc.nasa.gov/WWW/wind/valid/m6wing/m6wing01/m6wing01.html.

1.0 Importing Grid from PLOT3D File

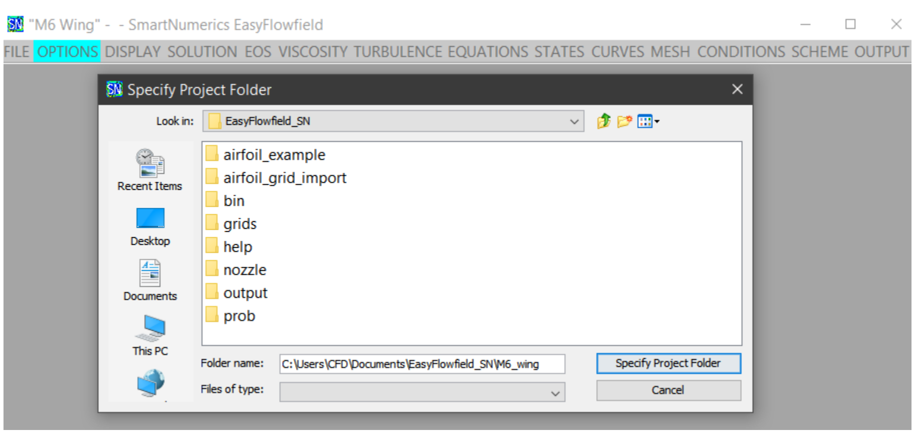

Fig. 1: Creating project folder for simulation of flow about M6 wing.

To prepare to use the grid, first open a Simulation Overview dialog under menu heading **OPTIONS** and change the title to "M6 Wing". Then click on Add Project Folder, navigate to the EasyFlowfield_SN folder, and append "M6_wing" to the folder name. You will be prompted to save the script file when you close the dialog. Please save the script as test.script in M6_wing/prob.

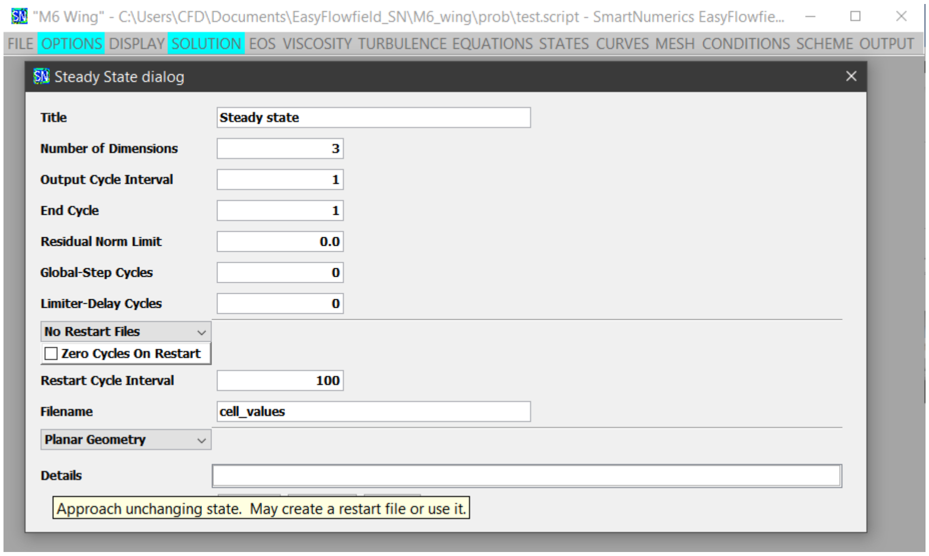

Fig. 2: Initial settings in Steady State dialog.

Next open the Steady State dialog under menu heading **SOLUTION**. Please set the number of dimensions to 3 and 'Output Cycle Interval' and 'End Cycle' to 1.

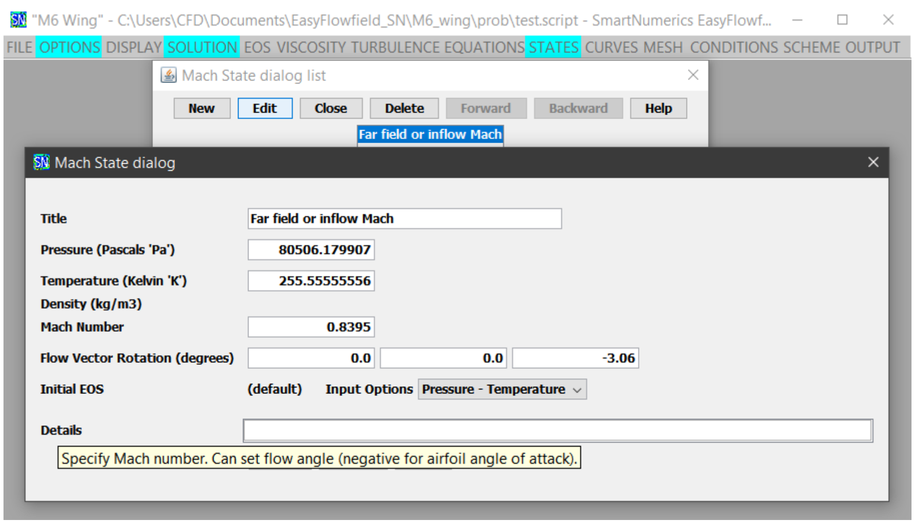

Fig. 3: Set values for initial condition and reference state.

Next open a Mach-State dialog under menu heading **STATES** and specify a pressure of 80506.179907 Pascals, a temperature of 255.55555556 degrees Kelvin, a Mach number of 0.8395, and a 'Flow-Vector Rotation' angle of -3.06 degrees about the z axis as displayed in the above figure. This state will provide the initial condition and act as a reference state for far-field and outflow boundary conditions.

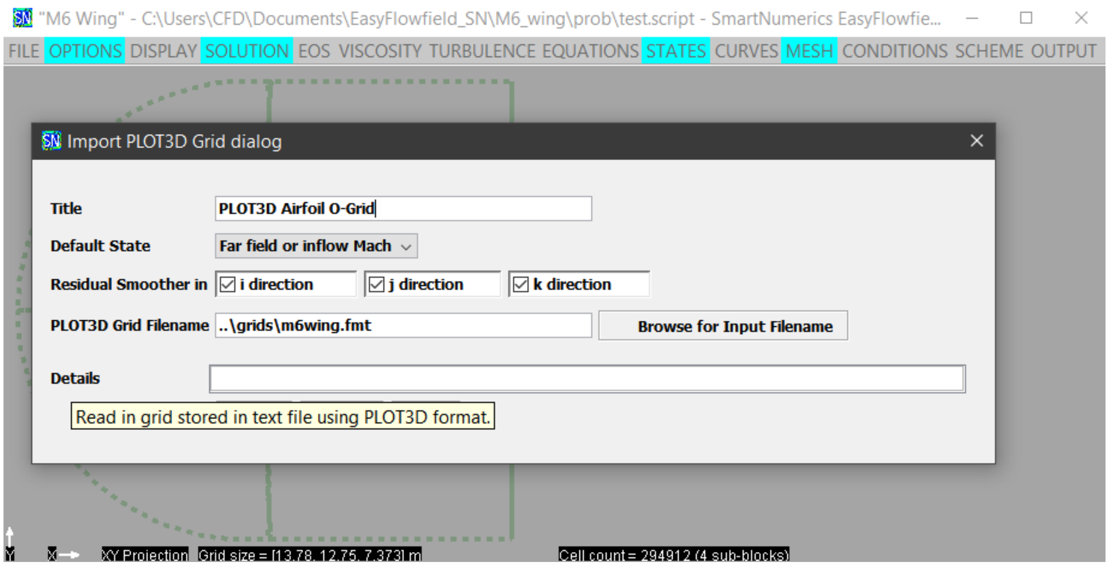

Fig. 4: Import PLOT3D grid.

Then download the PLOT3D grid file "m6wing.x.fmt" from the NASA website, rename it to "m6wing.fmt" and store it in M6_wing/grids. Next open the Import PLOT3D Grid dialog under menu heading **MESH**, change the title to "PLOT3D Airfoil O-Grid", click on ‘Browse for Input Filename’, highlight and select "m6wing.fmt". The grid will appear when you close the dialog. Loading may take a fair amount of time depending on the speed of your PC.

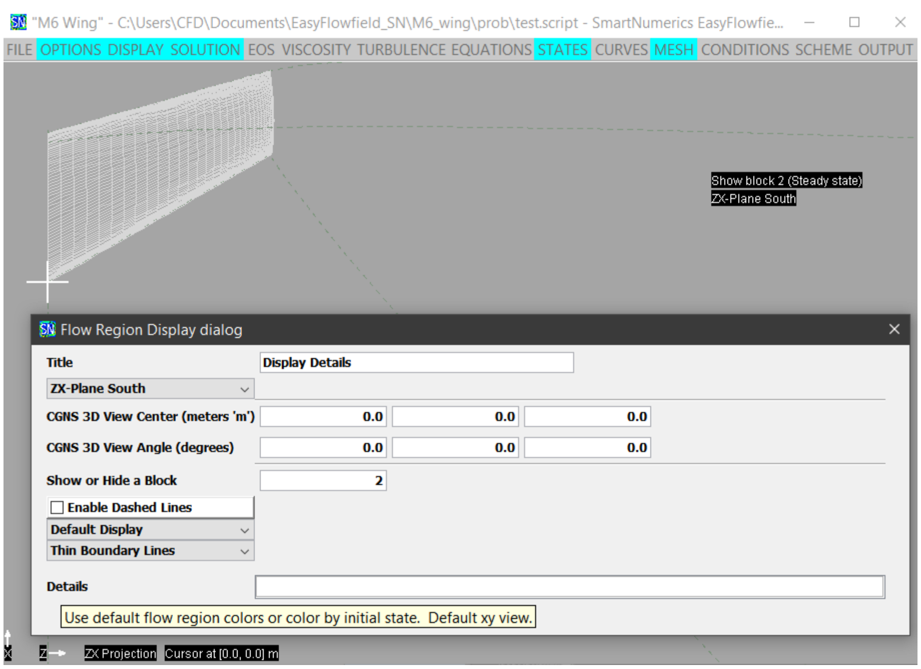

Fig. 5: Block 2 of imported grid for M6 wing viewed using Zoom mode.

This four-block grid is best viewed using thin boundary lines and the 'ZX-Plane south' option in the Flow Region Display dialog. Display of dashed lines should be disabled. The south face of Block 2 is displayed in the figure above. Other blocks can be displayed by pushing the + or - keys on your keyboard with the Flow Region Display dialog closed. Note that the chordwise direction is the *x* direction and the spanwise direction is the z direction. As detailed below boundary conditions must be added using four instances of the Basic Block Boundaries dialogs under menu heading **CONDITIONS**. The faces of adjacent blocks must be joined using several Secondary Interface dialogs.

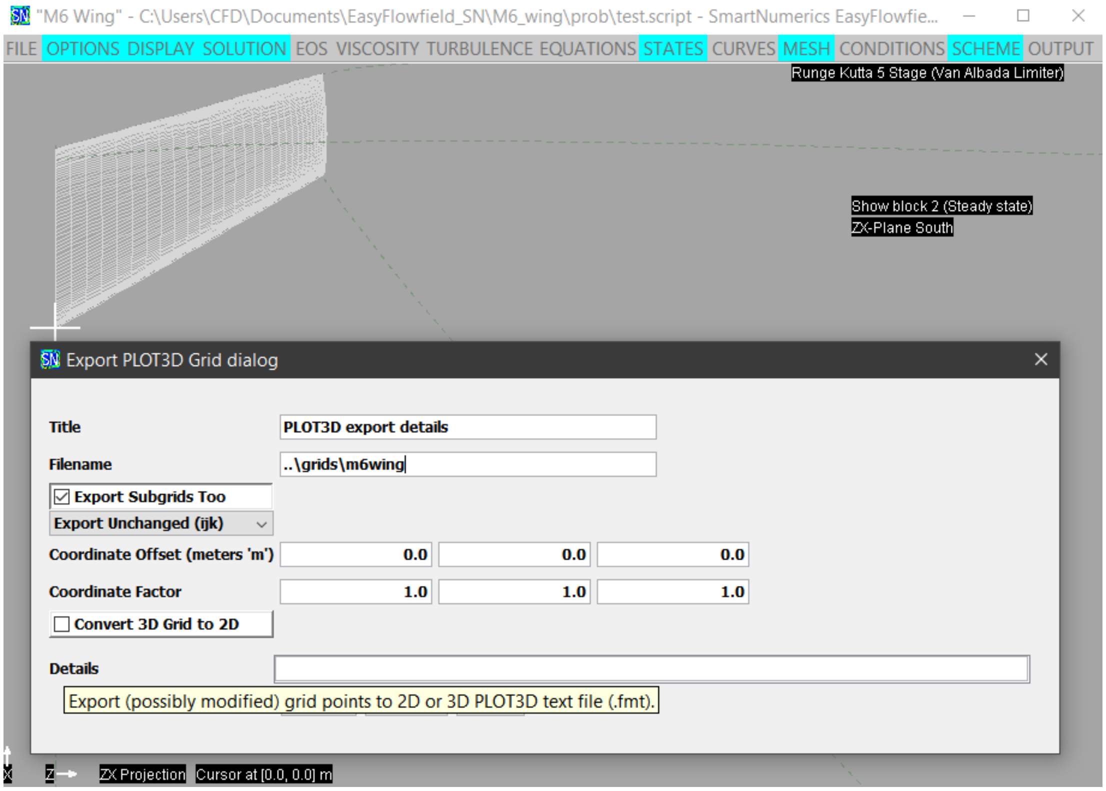

Fig. 6: Dialog used to export main grid and subgrid in PLOT3D format.

This is an exceptionally large grid and will probably visibly slow down operation of the GUI unless you have an extremely fast PC. You should create a coarser grid before proceeding further. You can use this smaller grid when learning how to connect the blocks and how to impose the boundary conditions. Open and close the Steady-State Solver dialog under menu heading **SCHEME**. Then open and close a Subgrid Startup Level dialog. The number of cells in the subgrid will be an eighth of the number in the original grid. Next, open the Export PLOT3D Grid dialog, activate 'Export Subgrids Too', and change the filename to "m6wing".

Please close the dialog, save the script, and run the simulation. The simulation will run for one cycle. The files m6wing_0.fmt and m6wing_1.fmt will be saved in M6_wing/grids. Please save test.script as test_orig.script.

Please reopen test.script, reopen the Import PLOT3D Grid dialog under menu heading **MESH**, click on Browse for Input Filename, highlight and select "m6wing_0.fmt". This grid will appear much more quickly after you close the dialog. Next open and delete the Export PLOT3D Grid dialog under menu heading **OUTPUT** to prevent an error message the next time that you run the simulation.

2.0 Preparation for Export of Grid to CGNS File

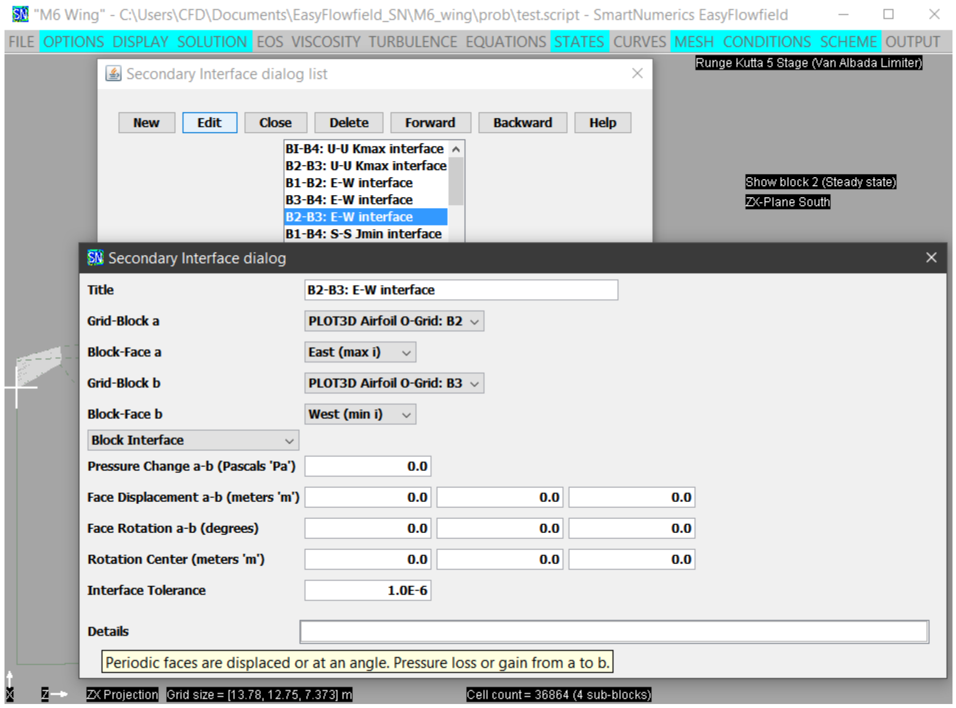

Fig. 7: Specify connections between block faces.

Next specify the block connectivity by creating several Secondary Interface dialogs under menu heading **CONDITIONS**. One such Block Interface is displayed in Figure 7. The connection has been summarized in the title for your convenience. Note that the 'Block Interface' option has been selected. Please connect the other block faces as indicated by the titles in the Secondary Interface dialog list displayed in the figure above. You should check your work by running a one-cycle interactive simulation every time you connect two blocks. The solver will halt with a mismatch error if the connection is incorrect.

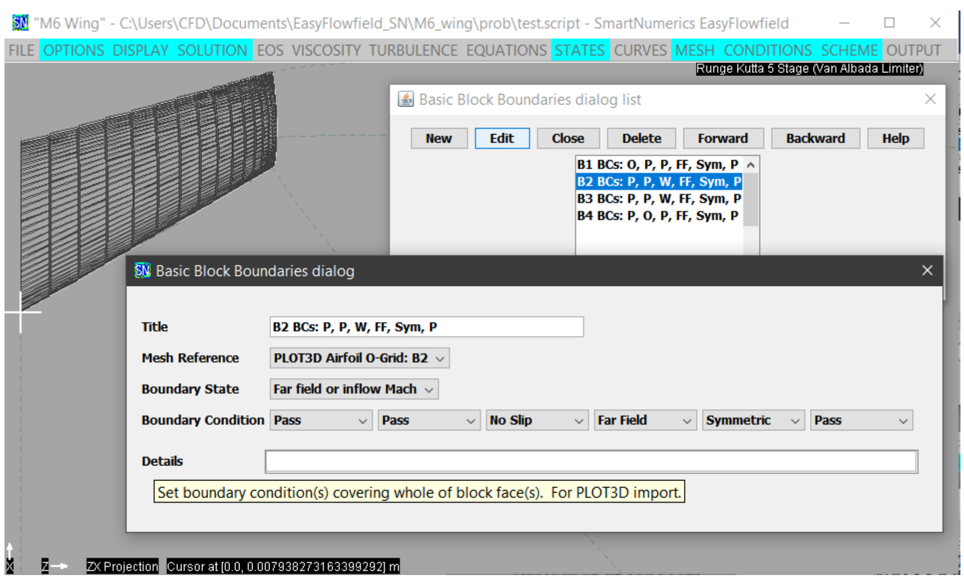

Fig. 8: Add appropriate boundary conditions.

Once the blocks have been correctly connected, add the appropriate boundary conditions. Figure 8 displays the boundary conditions for block 2. The boundary conditions have been summarized in the title using the sequence "P, P, W, FF, Sym, P" which stands for Pass, Pass, Wall, Far Field, Symmetry, and Pass. The simulation will involve turbulent flow, so a No-Slip boundary condition is used for the wall. Please specify the boundary conditions for the blocks as indicated in the Basic Block Boundaries dialog list displayed in Figure 8. Note that "O" stands for Outflow. Note that block faces connected by a block interface must both use the Pass boundary condition. **Please do not try to run the simulation before activating the turbulence model as detailed below.**

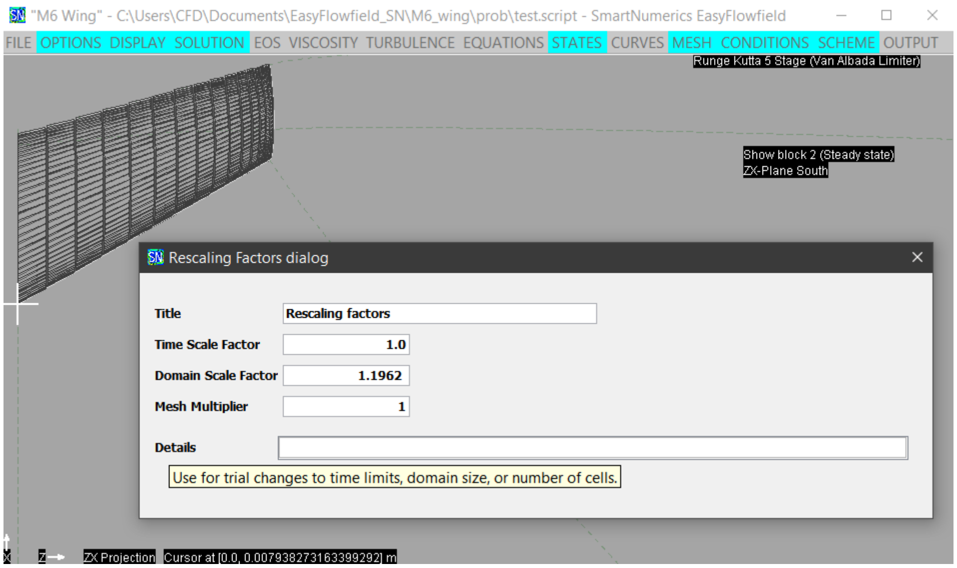

Fig. 9: Rescale grid.

The M6 validation test is based on the experimental measurements of Schmitt and Charpin [1]. The wind tunnel model had a half span of 1.1963 m and a mean aerodynamic chord of 0.64607 m. The root chord was 0.8059 m and the wing area was 0.7532 m^2. The grid has a half span of about 1.0165 m and a root chord of 0.6737 m. Both values are somewhat smaller than the values given for the wind tunnel model. To get the correct root chord the coordinates must be multiplied by 1.1962 using the Rescaling Factors dialog under menu heading **OPTIONS** as shown above.

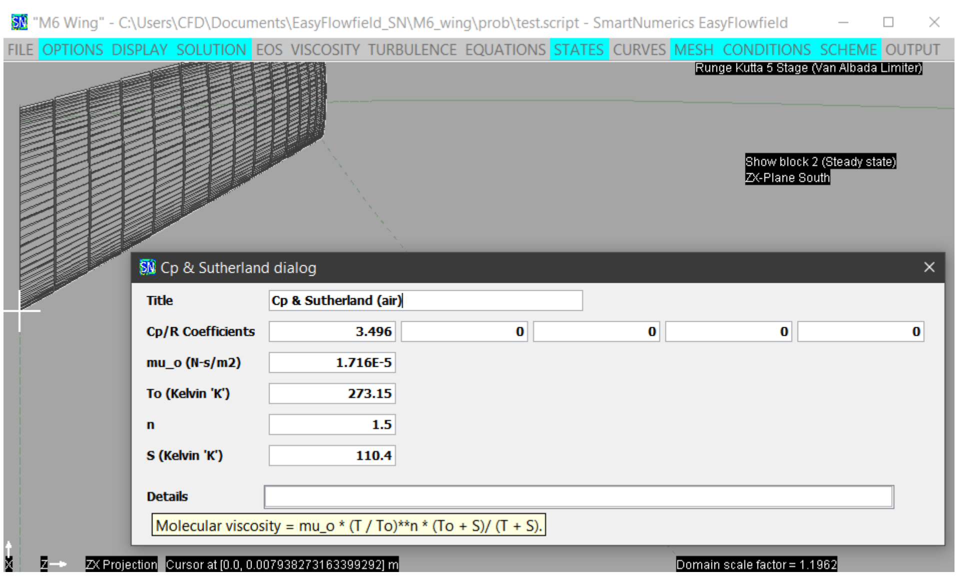

Fig. 10: Add Viscosity Model.

Since a No Slip boundary condition have been specified, a simulation involving laminar or turbulent flow must be performed. First, enable use of the Sutherland model for calculation of molecular viscosity by opening and closing the Cp & Sutherland dialog under menu heading **VISCOSITY**. The default parameters are for air.

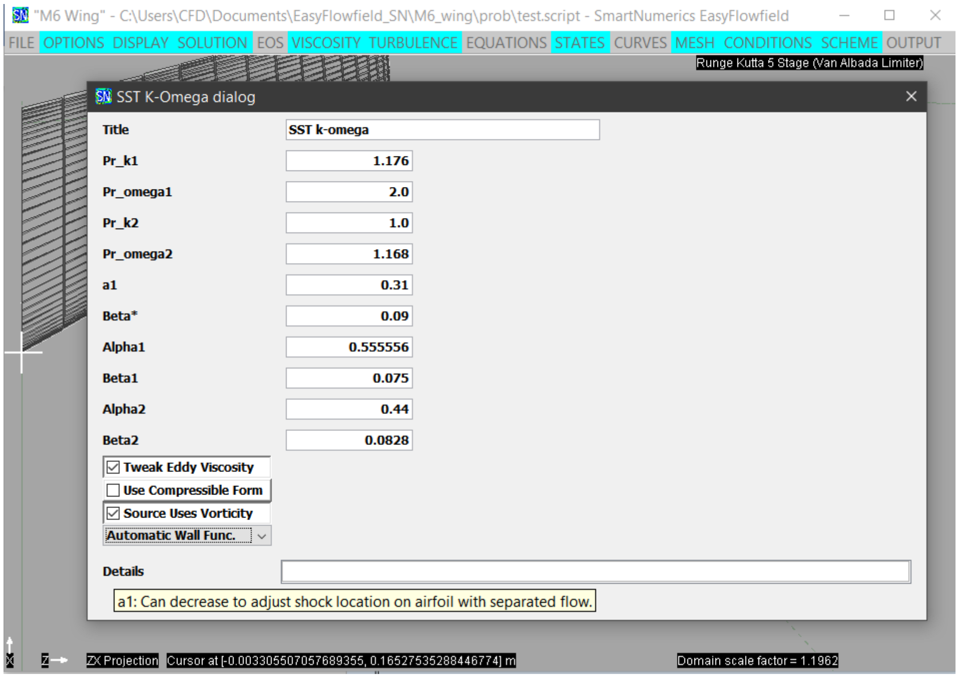

Fig. 11: Add use of a turbulence model.

Next open the SST k-Omega dialog under menu heading **TURBULENCE**. Select use of the automatic wall function since this grid is too coarse to model turbulent flow without a wall function.

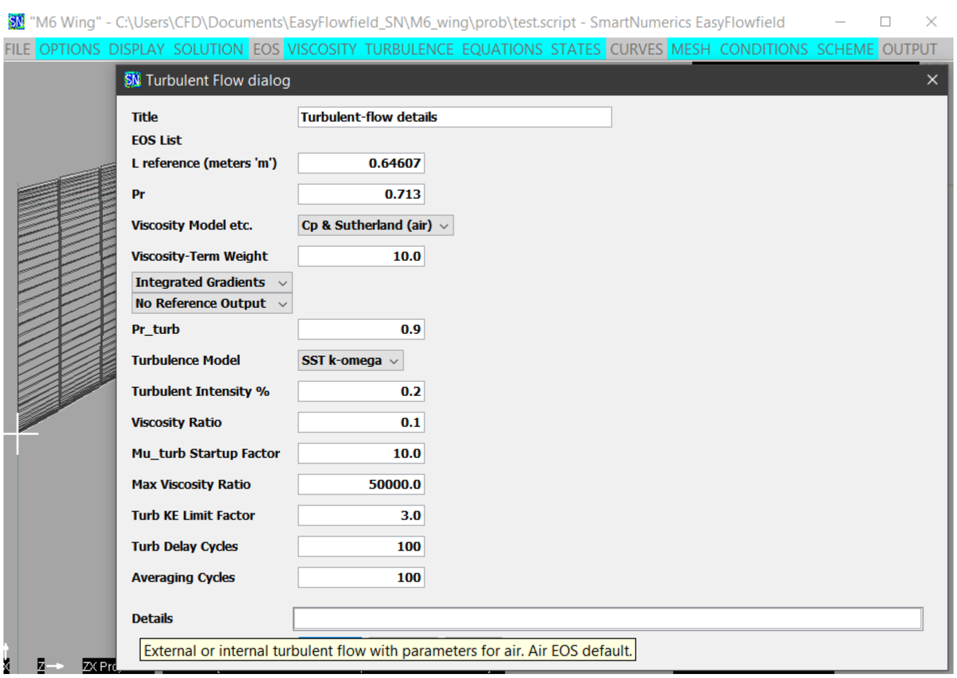

Fig. 12: Select parameters used with simulation of turbulent flow.

Then open the Turbulent-Flow dialog under menu heading **EQUATIONS** and set the reference length to the mean chord, 0.64607. The solver outputs the Reynolds number based on this length at the start of the simulation. Please also set ‘Viscosity-Term Weight’ to 10 which will reduce the time-step size in cells with a large value of eddy viscosity and has the effect of increasing stability as the simulation approaches the steady state while allowing larger time steps early in the simulation. The (far-field) turbulent intensity should be set to the experimental value of 0.2% and 'Turb KE Limit Factor' should be set to 3. The parameter ‘Max Viscosity Ratio’ has been set to 50,000 but the standard value of 10,000 would be satisfactory. The other parameters may be left at their default values. At this point you should try running the simulation for one cycle to ensure that the boundary conditions have been entered properly.

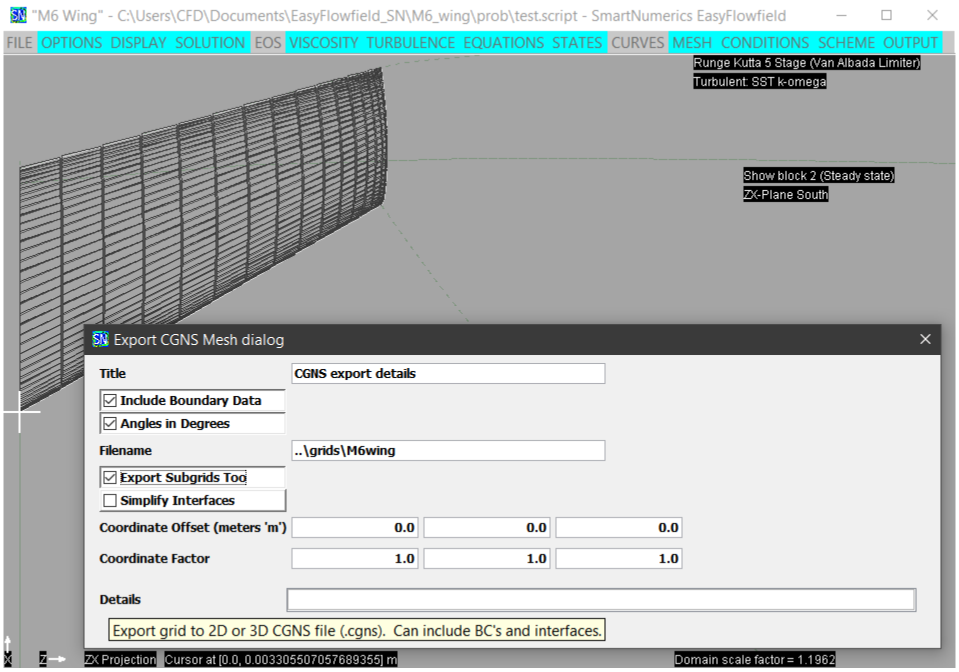

Fig. 13: Dialog used to export main grid and subgrids in CGNS format.

Please add two more Startup Subgrid Level dialogs. Next, open the Export CGNS Mesh dialog, activate 'Export Subgrids Too', and change the filename to "m6wing".

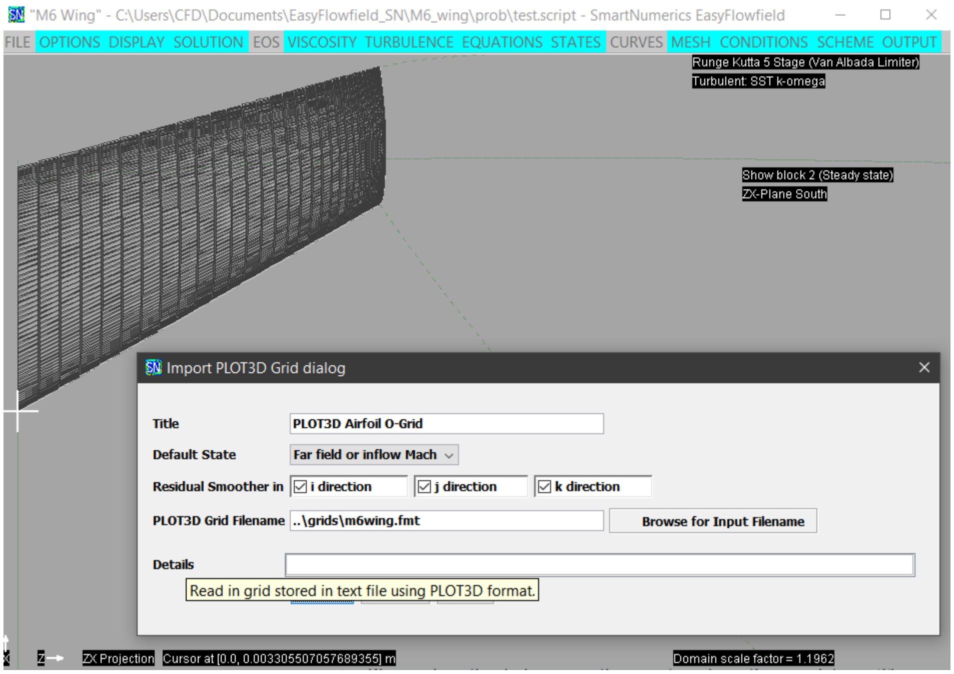

Fig. 14: Import the original grid.

Please reopen the Import PLOT3D Grid under menu heading **MESH**, click on Browse for Input Filename, highlight and select "m6wing.fmt". This fine grid will appear more slowly after you close the dialog. Please close the dialog, save the script, and run the simulation. The simulation will run for one cycle. The files m6wing_0.cgns through m6wing_3.cgns will be saved in M6_wing/grids. Note that a CGNS file is about half the size of a PLOT3D file for the same grid. Please save test.script as test_CGNS_export.script, in case you wish to modify certain boundary conditions at a later date.

3.0 Initial Simulation Using Coarse Grid

Please re-open test.script and delete the Import PLOT3D Grid dialog, open the Import CGNS Mesh dialog, and import m6wing_2.cgns which has one eighth of the cells in the finest grid contained in m6wing_3.cgns. Next delete all the dialogs under menu heading **CONDITIONS** since they are no longer needed. Also delete the Export CGNS Mesh dialog under menu heading **OUPUT** to prevent an error message when you run the simulation. The Rescaling Factors dialog under menu heading **OPTIONS** must also be deleted because the mesh contained in the CGNS file now has the correct root chord.

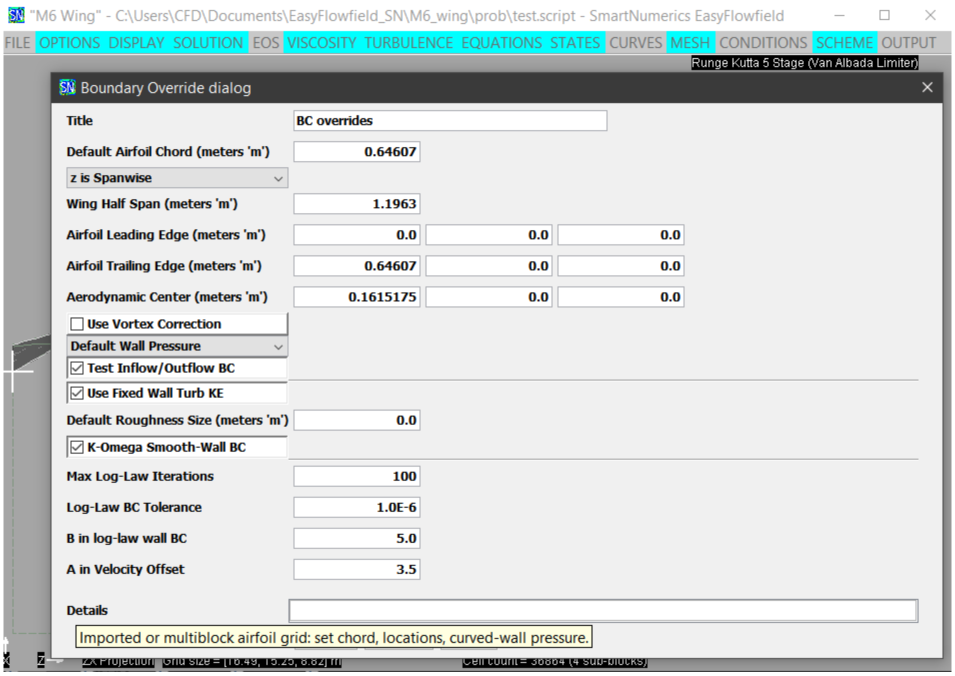

Fig. 15: Enter initial wing parameters.

Please open the Boundary Override dialog under menu heading **OPTIONS**. In the above figure, the default chord has been set to the mean aerodynamic chord of the wind-tunnel model and the wing half span has been set to the model half span. The distance between the airfoil leading and trailing edge must be equal to the chord. In this case, the leading edge has been arbitrarily set to 0 and the aerodynamic center has been set to one-quarter of the chord. A careful analysis would be required to determine the actual aerodynamic center. The location of the aerodynamic center only effects the calculation of the pitching moment and the vortex correction. In this case, the values of the aerodynamic center and the pitching moment will be corrected later and use of the vortex correction at the far-field boundary has been disabled. Note that the far-field and outflow boundaries are about eight chords from the wing surface. A discussion on the importance of and the approximate calculation the mean aerodynamic center and other wing parameters can be found in Phillips, Hunsaker, and Niewoehner [2].

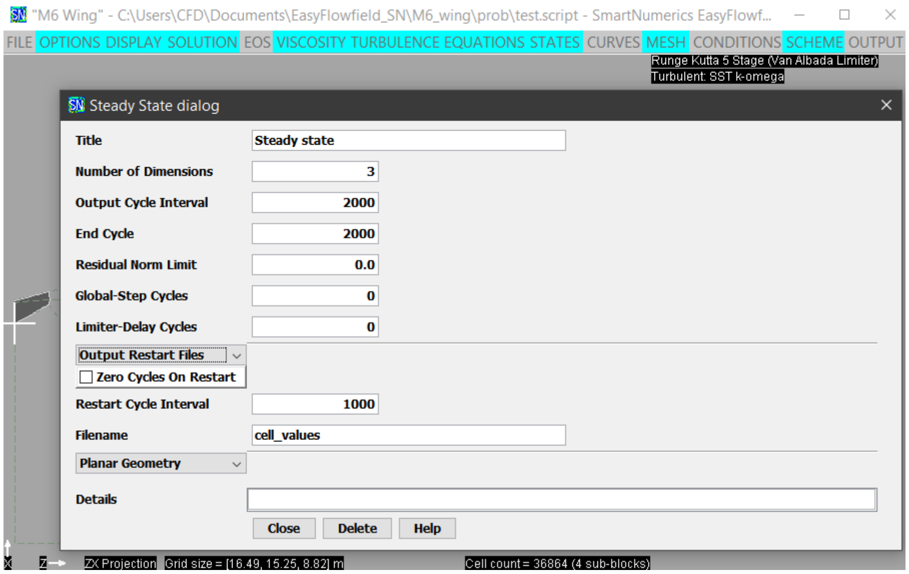

Fig. 16: Specify Output intervals etc. for simulation.

Please open the Steady State dialog under menu heading **SOLUTION** and set 'Output Cycle Interval' and 'End Cycle' to 2000. Also specify output of restart files with a 'Restart Cycle Interval' of 1000 cycles.

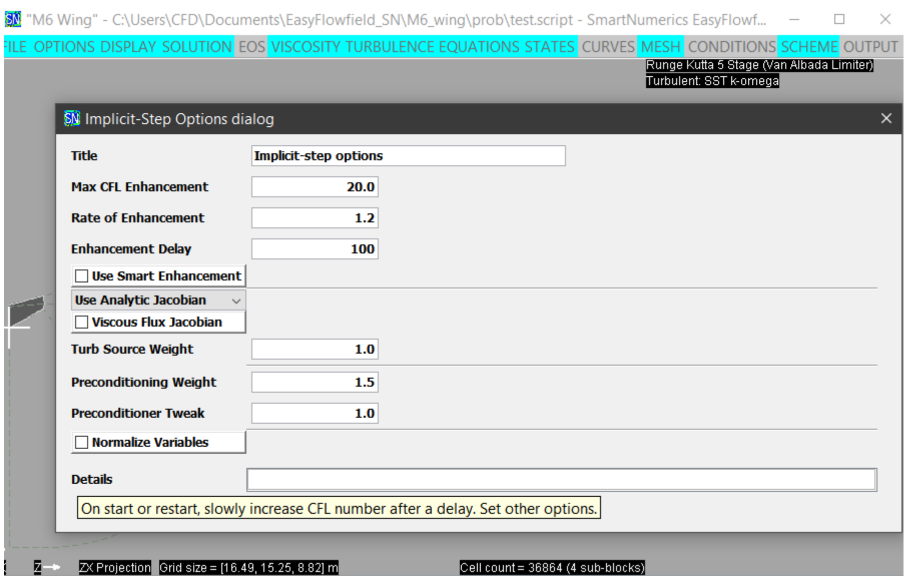

Fig. 17: Specify implicit-step options.

Next, open the Implicit-Step Options dialog and specify a maximum CFL enhancement of 20 after a delay of 100 cycles. After 100 cycles, the CFL number will by increased by a factor of 1.2 on each cycle until the initial CFL number has been increased by a factor of 20. Please disable use of smart enhancement.

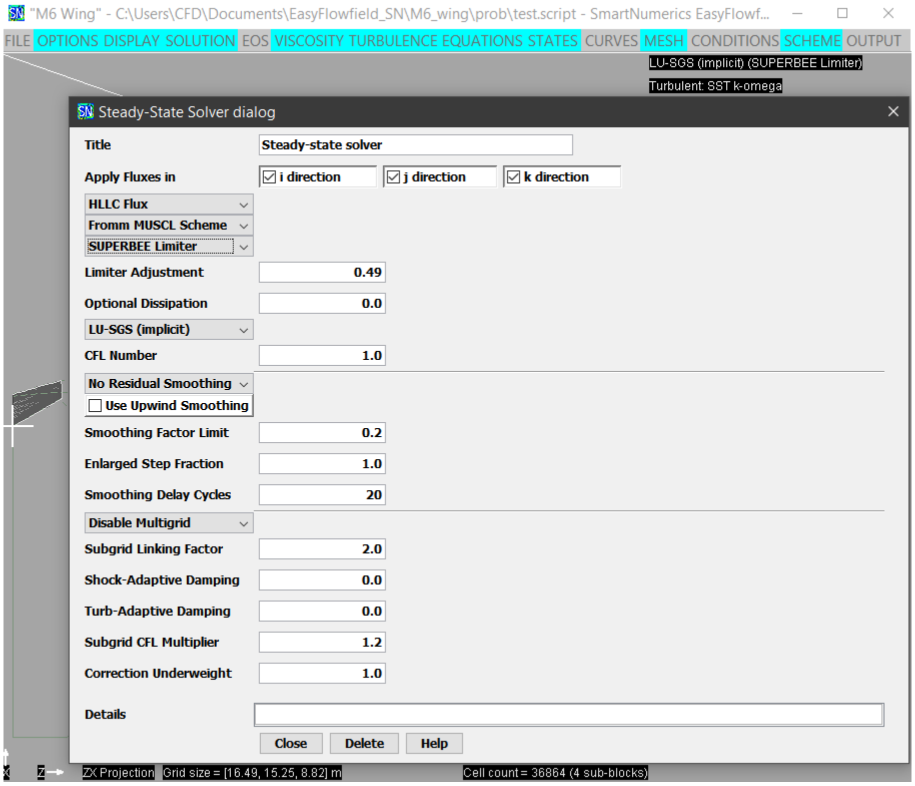

Fig. 18: Specify details of solution scheme.

Next, open the Steady-State Solver dialog under menu heading **SCHEME** and select the SUPERBEE limiter with a limiter adjustment of 0.49. Also select the LU-SGS solver, set the CFL number to 1, and disable multigrid. Please delete all the Startup Subgrid Level dialogs to avoid error messages or warnings when the simulation is run.

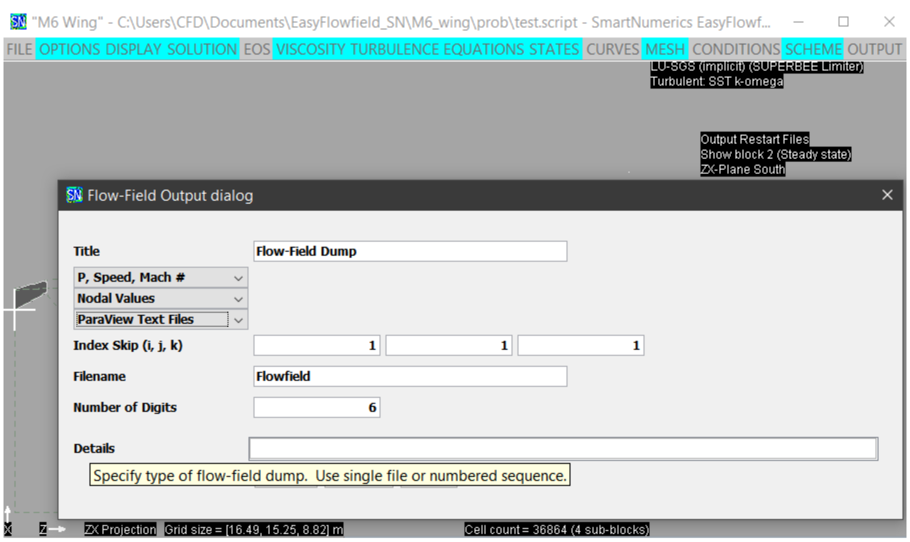

Fig. 19: Specify output for pressure and Mach contour plots.

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

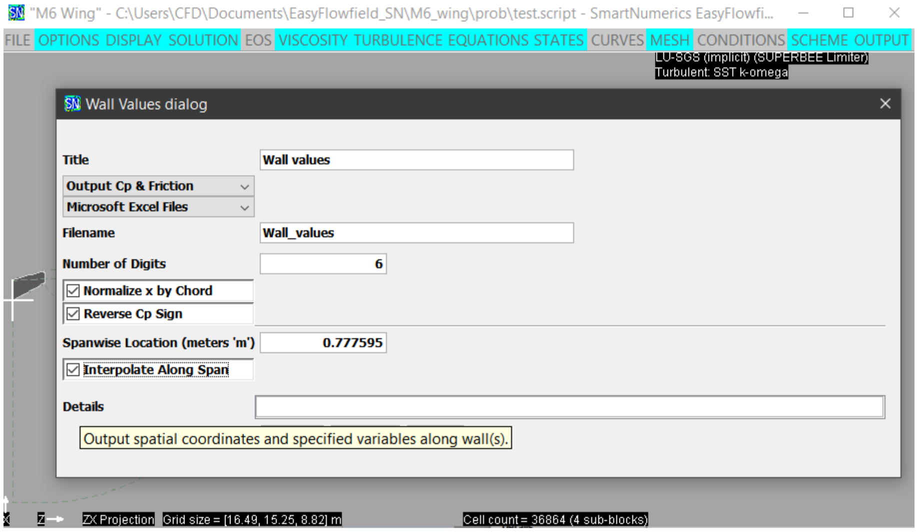

Fig. 20: Specify spanwise location etc. for output of pressure coefficients.

Next, open the Wall-Values dialog under menu heading **OUTPUT**, choose the Microsoft Excel file format, specify 'Output Cp & Friction', and activate normalization of wall coordinates by the chord. Also choose to reverse the Cp sign to match the experimental values contained in text files that can be downloaded from https://www.grc.nasa.gov/WWW/wind/valid/m6wing/m6wing.html. Then specify interpolation of values along the span and enter a spanwise location of 0.777595 meters which is 0.65 times the wing half span.

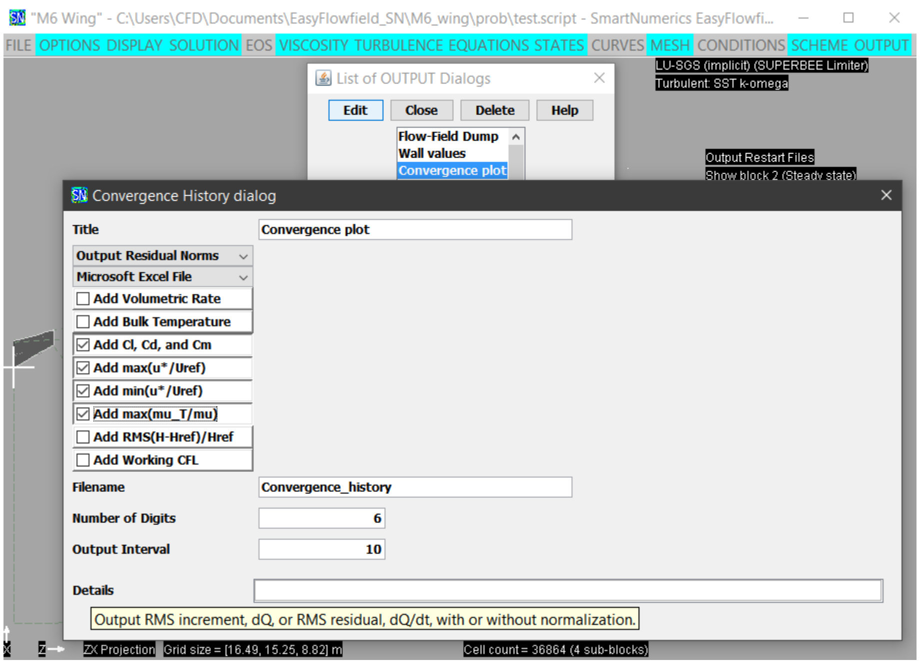

Fig. 21: Selecting measures of convergence.

Finally open the Convergence-History dialog and select use of the Microsoft Excel file format. Also select output of the airfoil force coefficients (Cl, Cd, Cm), the ratio of the minimum and maximum values of the friction velocity (u*) to the far-field velocity, and the ratio of the maximum eddy viscosity (μ_T) to the far-field molecular viscosity (μ).

Please close the dialog, save the script, and run the simulation. The ParaView and Excel output files will be saved in M6_wing/output. Please save test.script as test_coarse.script in case you want to rerun the simulation at a later date. The Reynolds number based on a mean aerodynamic chord of 0.64607 meters is output in the listing and should be 11.72×10^6.

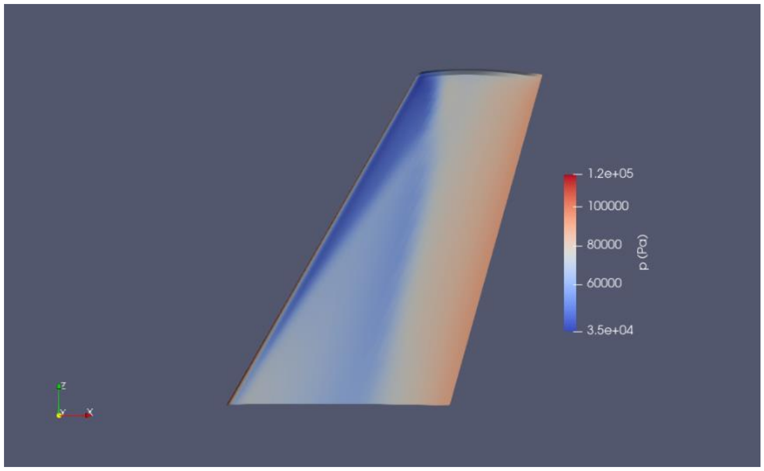

Fig. 22a: Pressure contours at 2000 cycles on M6 wing (from ParaView).

Figure 22a was plotted using ParaView. The file grid_block3_1.vts was loaded into ParaView and the 'Extract Subset' filter was applied with the upper j index reset to 0 and the representation set to 'surface'. If desired, contour lines can be obtained by then selecting the 'Contour' filter, selecting 'Compute Scalers', dividing the pressure isosurfaces value range into 24 values, setting representation to surface, setting colouring to 'Solid Color', using Edit to set the line colour to black, and specifying a line width of 4. New variables can be computed from a combination of existing variables using the Calculator filter.

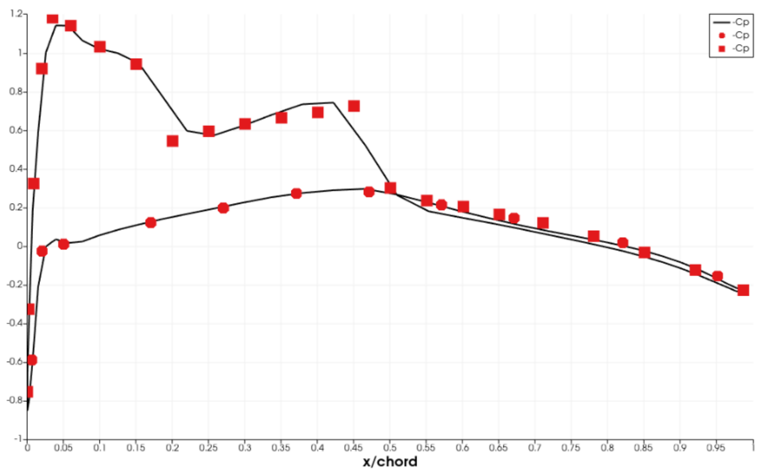

Fig. 22b: Cp at 65% of chord compared to experiment (coarse grid).

Figure 22b displays the Cp curve with reversed sign at 65% of span at 2000 cycles as a solid line. The circles mark the experimental values along the bottom of the wing and the squares mark the experimental values along the top of the wing. This plot was created by ParaView using an Excel file produced by the solver and two text files written in .csv format containing the experimental values. The experimental data files were created by adding commas and removing comments from text files originally written in Tecplot format. 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.

Schmitt and Charpin [1] measured values of the pressure coefficient (Cp) at various spanwise locations as detailed in the first column of Table 1. Also shown are the corresponding locations of the leading and trailing edge from the grid at each spanwise location. These were obtained by going into zoom mode clicking on the leading and trailing edge of the wing at the desired spanwise location. The local chord is the difference of the leading and trailing edge values. More accurate values of the local chord, trailing edge, and leading edge can be found in the listing when spanwise location is specified with output of 'Airfoil Cp & Friction' in the Wall-Values dialog.

Table 1: Spanwise location and corresponding leading and trailing edges.

Fraction of Half Span	Spanwise Location (m)	Leading Edge Position (m)	Trailing Edge Position (m)	Local Chord
0.20	0.239260	0.137686	0.870140	0.732454
0.44	0.526337	0.302740	0.950947	0.648207
0.65	0.777595	0.447202	1.021721	0.574519
0.80	0.957040	0.550375	1.072240	0.521865
0.90	1.076670	0.619157	1.105863	0.486706
0.95	1.136485	0.653548	1.122673	0.469125
0.99	1.184337	0.681061	1.136122	0.455061

The Cp values from the simulation in Figure 22b are plotted with respect to x normalized by the local chord 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 value of x normalized by local cord is computed using

$$x/\text{chord} = \frac{X - X_{\text{le}}}{X_{\text{te}} - X_{\text{le}}}$$

where X_{le} and X_{te} are respectively the local leading and trailing edge values and X is the x coordinate at the center of the no-slip cell face interpolated to the specified spanwise location.

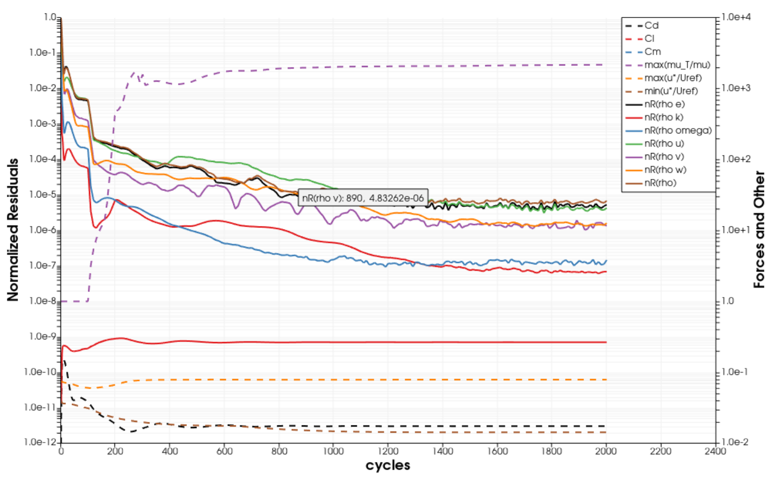

Fig. 22c: Convergence for turbulent flow about M6 wing (coarse grid).

Figure 22c displays the convergence history. Note how the airfoil lift and drag coefficients have reached constant values within 1000 cycles. The minimum and maximum values of friction velocity reach a constant value shortly thereafter and the maximum value of eddy viscosity has reached a constant value by 1200 cycles. The maximum normalized residual is less than 10^{-5} by 1500 cycles.

The final values of lift and drag are 0.265 and 0.0174, respectively. The pitching moment is -0.124 but this is computed with an incorrect aerodynamic center which can be corrected as described in the next section.

4.0 Finding the Mean Aerodynamic Center of the Wing

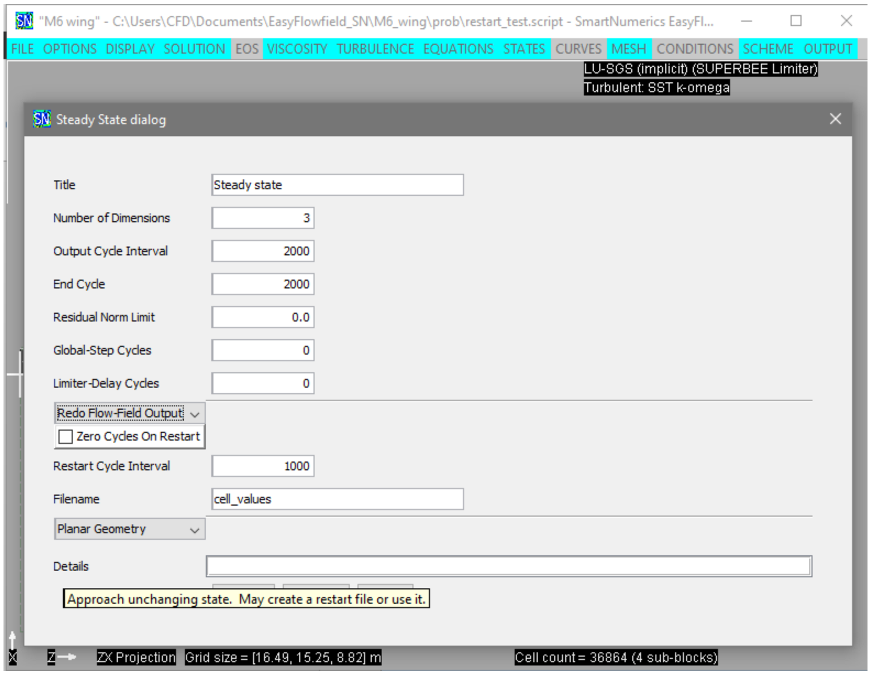

Fig. 23: Use of restart files to repeat output of flow-field and wall values.

First copy test.script to restart_test.script, open the Steady State dialog under menu heading **SOLUTION**, and change ‘Output Restart Files’ to ‘Redo Flow-Field Output’. Please close the dialog and save the script.

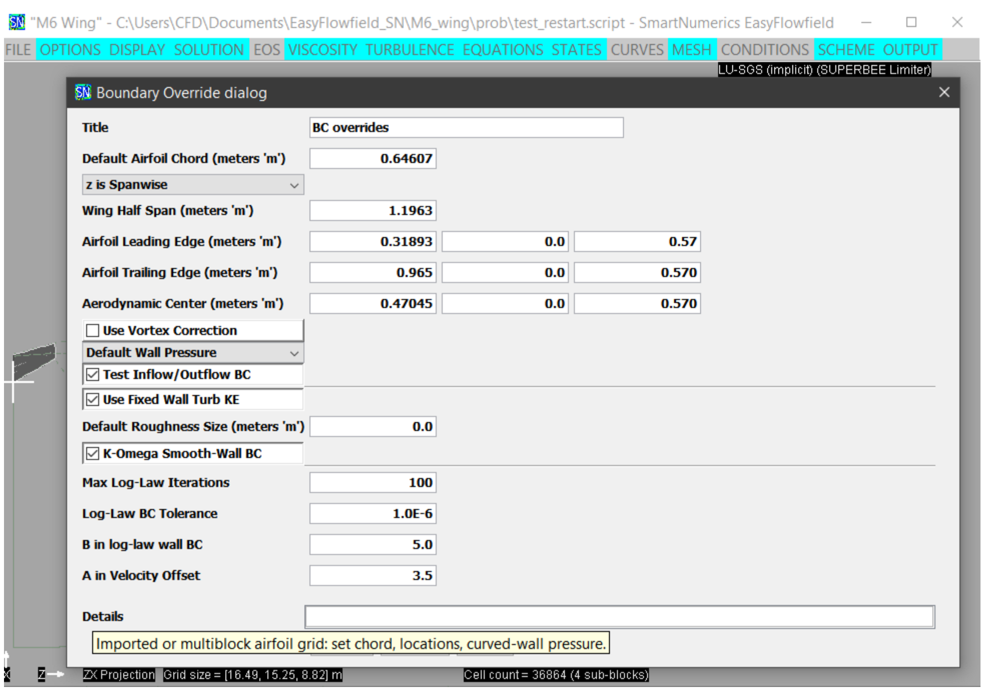

Fig. 24: Improved definition of wing.

To obtain a first approximation to the mean aerodynamic center, please change certain entries in the Boundary Override dialog as indicated in the above figure. That is, set the trailing edge coordinate to the leading-edge coordinate on the wing where the local chord is equal to the experimental value of mean aerodynamic chord. In this case, the chord at z = 0.57 meters is approximately equal to the mean aerodynamic chord and the tailing edge coordinate is approximately at x = 0.965 meters. The leading-edge coordinate has been set to the trailing edge coordinate plus the default chord. The distance between the specified leading edge and the specified training edge must be equal to the default cord or the solver will produce an error message. The aerodynamic center has been set to the quarter chord point at z = 0.57. Note that only the first two components of the aerodynamic center will affect the pitching moment and will not affect the solution as long as a vortex correction is not applied to the far-field boundary. That is, the spanwise value of 0.57 has no effect on the simulation or the pitching moment. In fact, you can set the spanwise component of the leading edge, trailing edge, and the aerodynamic center to the wing root location or symmetry plane (z = 0) without effecting this simulation.

Please close the dialog, save restart_test.script, and run the simulation by clicking on **Background Simulation** under menu heading **FILE**. This script uses the restart files created by the previous simulation. **Do not click on the Automated Simulation option since that will re-execute test.scrpt.**

The final value of pitching moment in the listing will be -0.00328. A wing with zero camber such as the M6 wing should have a pitching moment of zero about the aerodynamic center for low values of AOA if the far-field Mach number is not too high. An alternative procedure will be described later. The optimum chordwise location of the aerodynamic center can be quickly determined by a process of trial and error. **For the purposes of this tutorial, I will proceed as if the Mach number is not too high.**

Please change the chordwise location of the aerodynamic center to 0.461 meters and re-run restart_test.script using Background Simulation. The final value of pitching moment in the listing is then -0.000606. As shown later, the pitching moment computed using a finer grid will differ slightly.

Finally save restart_test.scipt as test.script, open the Steady State dialog under menu heading **SOLUTION** and specify ‘Output Restart Files’. Next, open the Boundary Override dialog under menu heading **OPTIONS** and activate 'Use Vortex Correction'. Please close the dialog, save the script, and run the simulation by clicking on Automated Simulation under menu heading **FILE**. The solver will perform the simulation specified by test.script then redo the output as specified in restart_test.script. Please save test.script as test_coarse_final.script in case you want to re-run the simulation later.

The final values of lift and drag are 0.263 and 0.0173, respectively. The pitching moment is -0.000555. Application of the horseshoe vortex correction has caused a negligible change in the force coefficients. The convergence history has not been appreciably affected as seen in Figure 25.

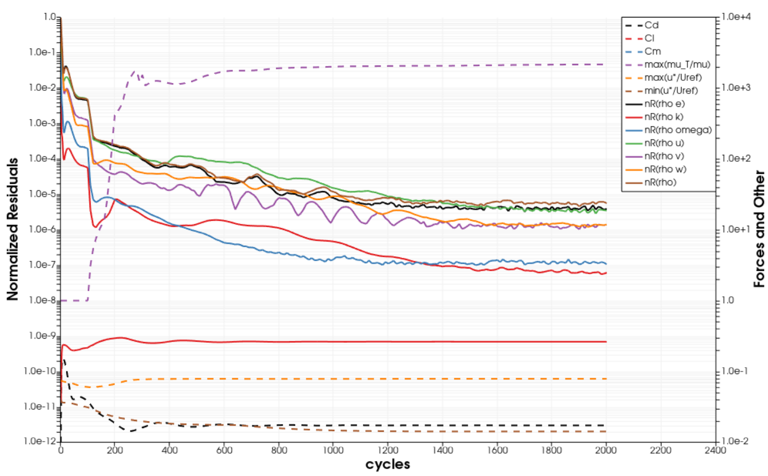

Fig. 25: Convergence for turbulent flow about M6 wing (coarse grid, vortex correction).

The value of the spanwise coordinate of the aerodynamic center was based on a guess. This value is not used in the horseshoe vortex correction and thus has no effect on the solution. Phillips, Hunsaker, and Niewoehner [2] provide expressions for the spanwise location of the aerodynamic center.

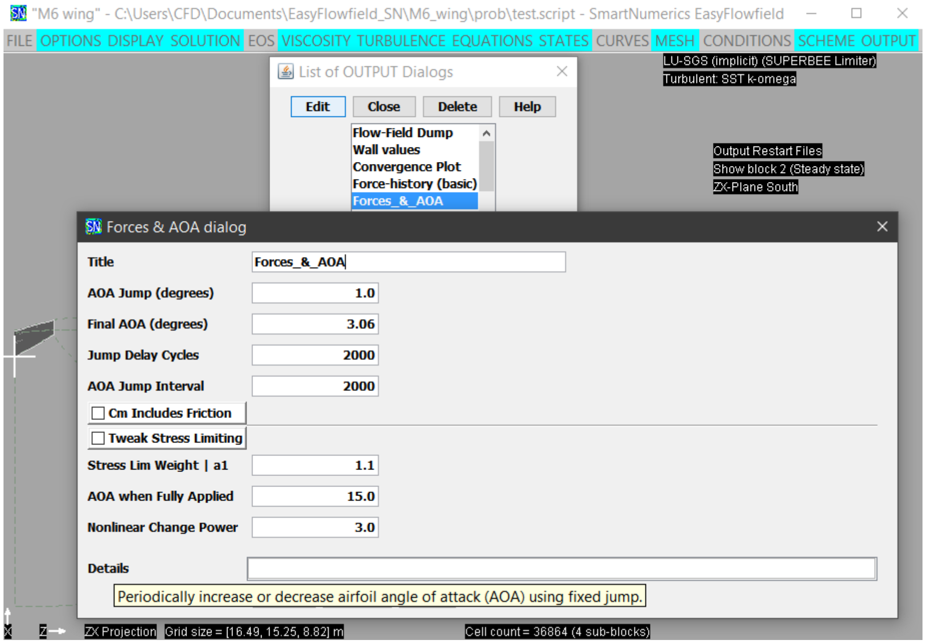

Fig. 26: Dialog used to impose periodic jump in AOA.

A slightly more complicated procedure is required to find the aerodynamic center if the Mach number is too large or if the wing has a nonzero camber. A wing or airfoil with nonzero camber will have a nonzero value of pitching moment about the aerodynamic center at all angles of attack However, the aerodynamic center has the property that the pitching moment about the aerodynamic center will change only slightly as the angle of attack is changed. Thus, you can determine the location of the aerodynamic center by running the simulation with several values of AOA. This can be most easily achieved by using the Forces and AOA dialogs under menu heading **OUTPUT**. In the above figure, the Forces and AOA dialog has been configured so that AOA is increased by one degree after a delay of 2000 cycles. The AOA is subsequently increased by one degree every 2000 cycles until 3.06 degrees is reached. The initial flow angle must be set to 0.94 degrees (AOA = -0.94) and 'End Cycle' in the Steady State dialog under menu heading **SOLUTION** must be set to 10000. Restart files must be output every time the AOA is changed. The listing will contain a sequence of estimates of the aerodynamic center. After the simulation has been performed, restart_test.script can be used to redo output using the different estimates of the aerodynamic center. The aerodynamic center is discussed in detail in the validation document for turbulent flow about airfoils. **After, you finish this tutorial, you should perform this procedure using the current coarse grid. You will find that a slightly better estimate for the aerodynamic center is output for AOA 2.06 degrees. This is [0.459043, 0.0245763, 0.57] m. The aerodynamic center estimated for AOA 3.06 degrees will result in a less satisfactory pitching-moment coefficient curve due to the presence of relatively strong attached shocks. If you then try a Mach number of 0.6395, the aerodynamic centers specified for 2.06- and 3.06-degrees AOA will both result in satisfactory pitching-moment coefficient curves. The pitching-moment coefficients over the 4-degree range of AOA will deviate from 0 by less than -0.00045.**

5.0 Simulation Using Finest Grid

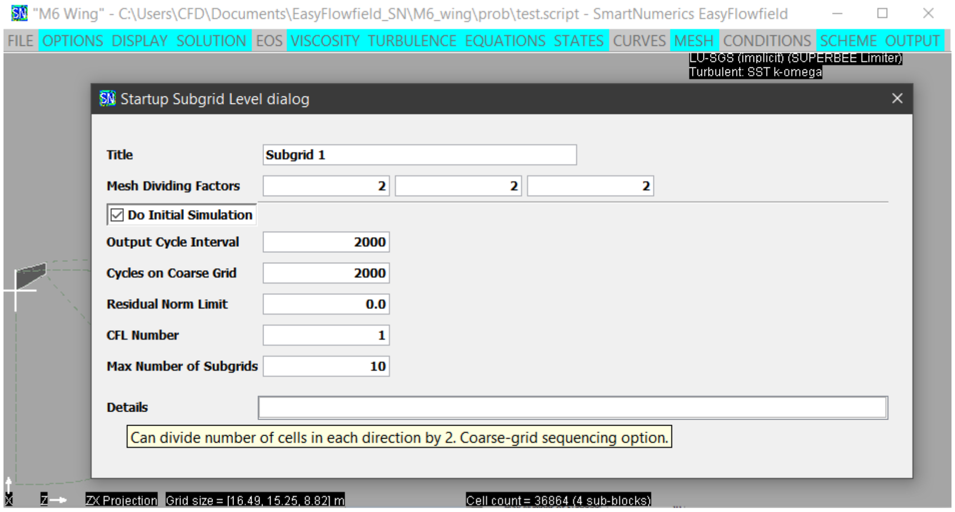

Fig. 27: Set up initial simulation on subgrid.

Please create a Startup Subgrid Level dialog in test.script and specify ‘Do Initial Simulation’. Also set ‘Output Cycle Interval’ and ‘Cycles on Coarse Grid’ to 2000. Finally set the CFL number to 1.

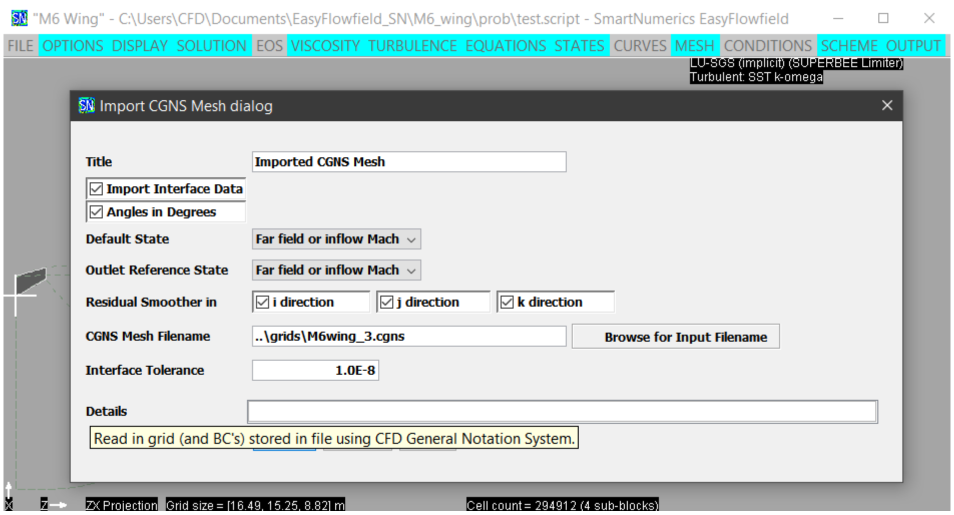

Fig. 28: Import finest grid from CGNS file.

Next open the Import CGNS Mesh dialog under menu heading **MESH** and select the "m6wing_3.CGNS" file. Please close the dialog and save test.script.

Then save test.script as restart_test.script, open the Steady State dialog and select 'Redo Flow-Field Output'. Please close the dialog and save restart_test.script.

Please reopen test.script and run the simulation by clicking Automated Simulation under menu heading **FILE**. After 2000 cycles, the solution using the coarse subgrid will be transferred to the main grid which has eight times the number of cells. The simulation on this grid will take 2000 additional cycles as specified in the Steady State dialog and will require approximately eight times the execution time used for the coarse grid. The output will be redone using restart_test.script after the execution of the simulation specified by test.script is complete.

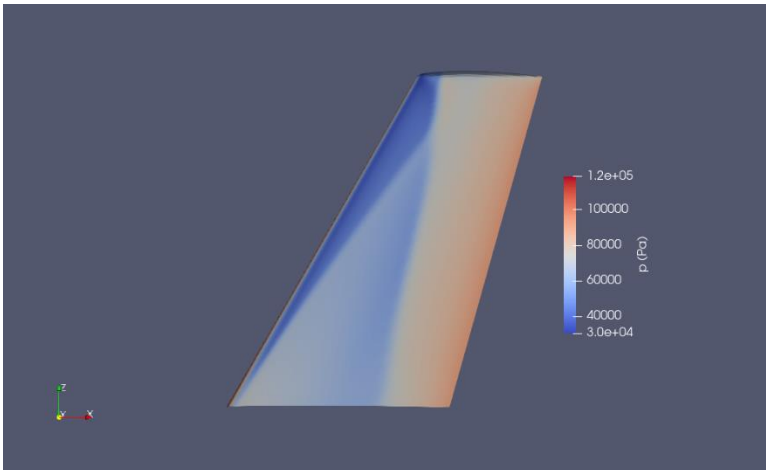

Fig. 29a: Pressure contours at 4000 cycles on M6 wing (from ParaView).

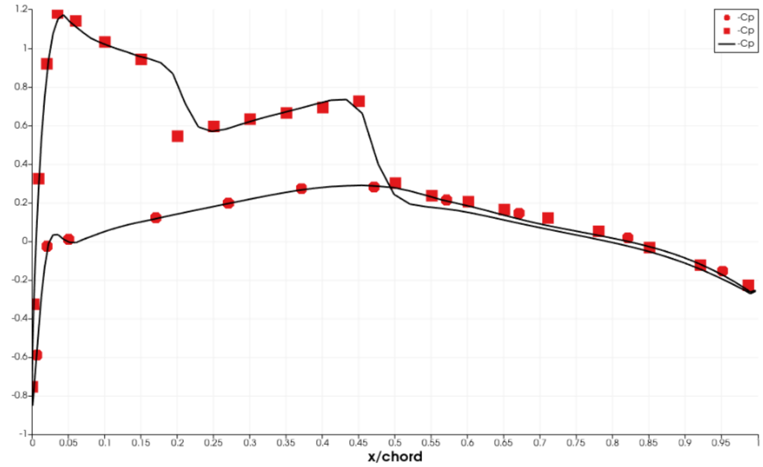

Fig. 29b: Cp at 65% of chord at 4,000 cycles (finest grid).

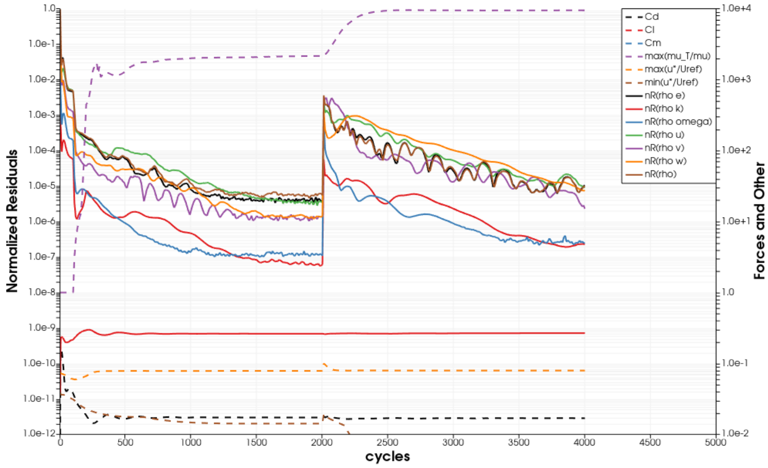

Fig. 29c: Convergence for turbulent flow about M6 wing (coarse and finest grid, vortex correction).

Figure 29a through 29c display the results of the simulation. As in the previous simulation, the lift and drag on the coarse grid are 0.263 and 0.0173, respectively. The pitching moment is -0.000555

The lift and drag on the fine grid are 0.267 and 0.0170, respectively. The pitching moment is 0.00107. Trial and error using restart_test.script to vary the mean aerodynamic center indicates that a value of 0.459 meters results in a pitching moment of 0.000237 on the fine grid and -0.00137 on the coarse grid.

Cp curves at a different spanwise location can be plotted without repeating the simulation by using restart_test.script. To do this, please open the Wall Values dialog under menu heading **OUTPUT** and specify a different spanwise location taken from Table 1.

References

[1] Schmitt, V., Charpin, P., "Pressure distributions on the ONERA-M6-Wing at transonic Mach numbers," in Experimental Data Base for Computer Program Assessment, AGARD Report AR 138, 1979.

[2] Phillips, W. F., Hunsaker, D. F., and Niewoehner, N. R., "Estimating the subsonic aerodynamic center and moment components for swept wings," Utah State University, Mechanical and Aerospace Engineering Faculty, 2008.