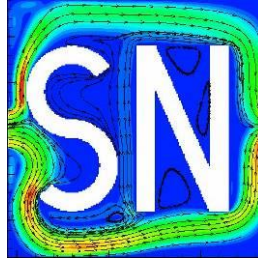


EasyFlowfield Tutorial 2: Flow About an Airfoil Using Imported 2D and 3D Grids
SmartNumerics Simulation Solutions Inc.



Version 6
June 4, 2020

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

This tutorial covers the basics of exporting and importing grids using the CGNS or PLOT3D formats. Grid import is typically used to obtain structured grids more complex than those created internally by EasyFlowfield.

For this exercise, you must first create some grid files to import. First use Microsoft Windows File Explorer to copy the airfoil_example folder from Tutorial 1. Save the copy as airfoil_grid_import. Use the GUI to open the test_implicit.script file in the airfoil_grid_import/prob folder. Use Save Script As to save it as test.script. Save the script one more time as test_export.script.

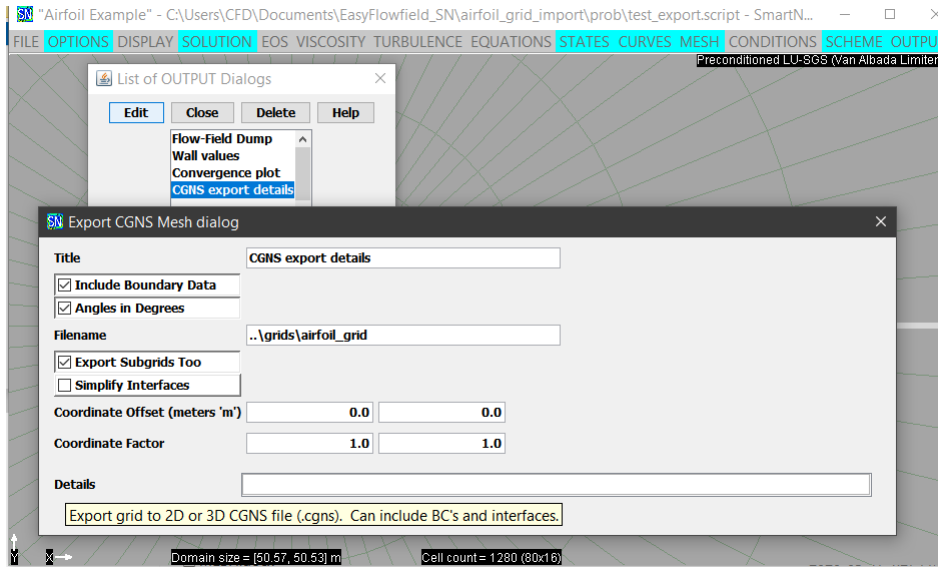


Fig. 1: Dialog used to export a grid file using the CGNS format.

Next open the Export CGNS Mesh dialog under menu heading **OUTPUT**. Change the name of the output file to "airfoil_grid" and activate the 'Export Subgrids Too' checkbox.

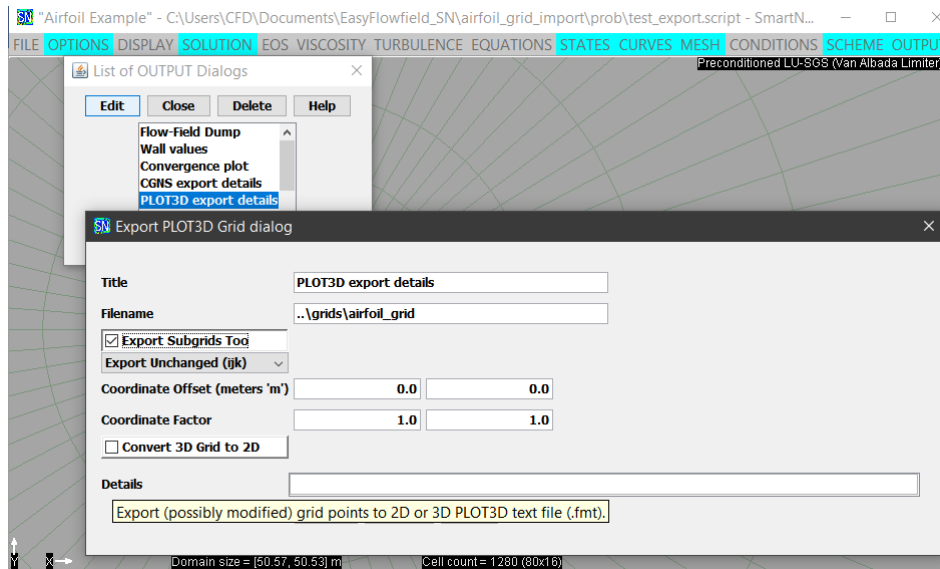


Fig. 2: Dialog used to export a grid file using the PLOT3D format.

Next open the Export PLOT3D Grid dialog. Change the name of the output file to "airfoil_grid" and activate the 'Export Subgrids Too' checkbox'.

Reopen the Steady State dialog under menu heading **SOLUTION** and set 'Output Cycle Interval' and 'End Cycle' to 1. Then reopen both Fine-Grid Transfer dialogs under menu heading **SCHEME** and set 'Output Cycle Interval' and 'Additional Cycles' to 1.

Save the script and run the simulation using the 'Background Simulation' option under menu heading **FILE**. This option runs the currently open script which is test_export.script. The CGNS grid files airfoil_grid_0.cgns through airfoil_grid_5.cgns and the PLOT3D files airfoil_grid_0.fmt through airfoil_grid_5.fmt will be saved in the airfoil_example/grids folder. These files store the node coordinates of the three subgrids, the initial grid, and the two fine grids. Details of their generation will be included in the listing.

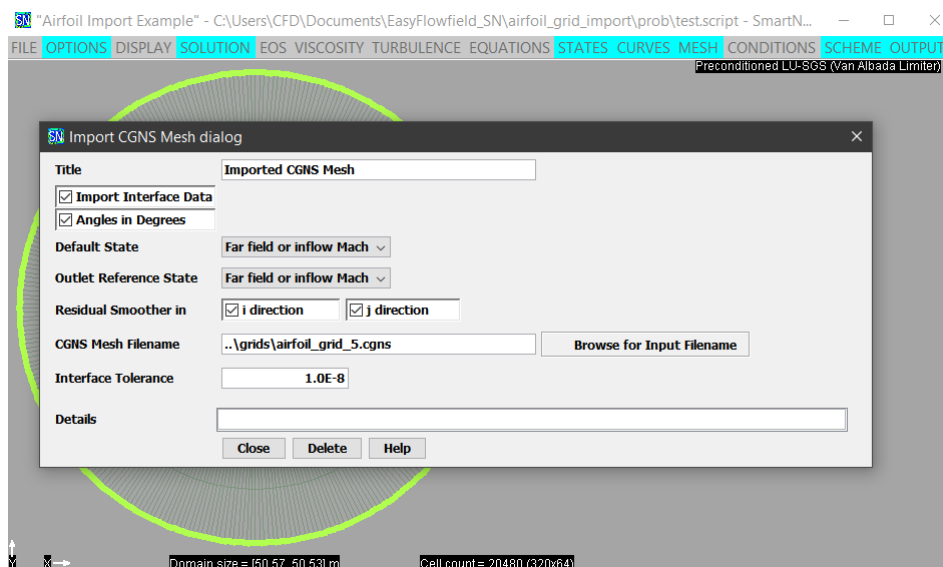


Fig. 3: Dialog used to import grid file in CGNS format.

Next open the test.script file. Open the Simulation Overview dialog under menu heading **OPTIONS** and change the title to “Airfoil Import Example”. Please open the O-Grid dialog under menu heading **MESH** and delete it. Next open the Import CGNS Mesh dialog, click on Browse for Input Filename and select "airfoil_grid_5.cgns". The grid will appear after the dialog is closed. Note that the boundary conditions are also imported from the CGNS file and the entire grid is displayed including the far-field boundary (yellow). Before proceeding further, please open the Airfoil Outline dialog under menu heading **CURVES** and select ‘Show Curve Points’.

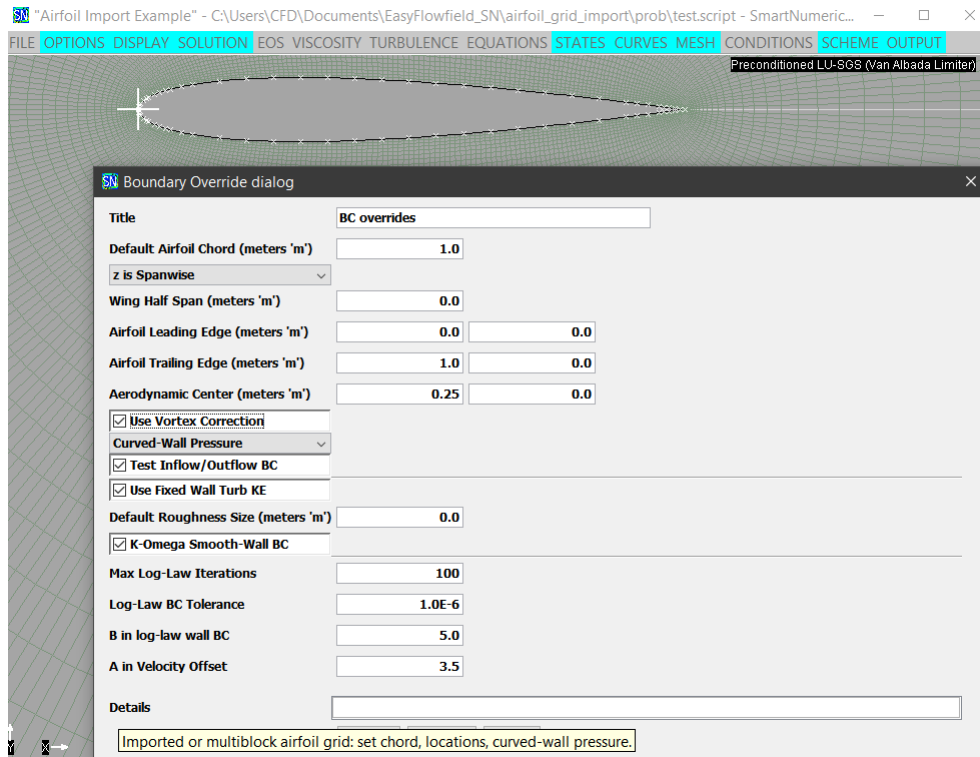


Fig. 4: Setting airfoil chord, leading edge, trailing edge, quarter-chord point and other parameters.

You must provide additional information, if flow about an airfoil or wing is being simulated using an imported grid. Please open the Boundary Override dialog under menu heading **OPTIONS** and specify the default chord (1 m), the leading edge coordinates ([0,0] m), the trailing edge coordinates ([1, 0] m), and the aerodynamic center ([0.25, 0] m) which is normally set at the quarter chord point. Also select use of vortex correction at the far-field boundary and curved-wall calculation of pressure at the airfoil surface. These values and options are selected to match those used by the solver in the previous example with an internally generated o-grid. In Figure 4, the Slip boundary is marked by a black line and the wake line where two grid faces meet at marked by a white line indicating a Pass boundary condition. The two adjacent grid faces are joined by a block interface that is also stored in the CGNS file.

In Figure 4, Zoom Mode, documented in help under menu heading **MESH**, has been used to allow inspection the airfoil surface and to verify the locations of the leading and trailing edges. The match between the original Airfoil Outline points marked with X marks and the airfoil surface marked by a black Slip boundary-condition line is best determined by opening the Flow Region Display dialog under menu heading **DISPLAY** and replacing the 'Default Boundary Lines' option by the 'Thin Boundary Lines' option. The input value of default chord controls the plotted size of the Airfoil Outline. Please hit the ‘Esc’ key on your keyboard when you wish to get out of zoom mode.

Next open and delete both Fine-Grid Transfer dialogs under menu heading **SCHEME**. Since this is an imported grid, it is not possible to make the grid finer. The sequence of grids of increasing fineness can be obtained by doing the first two simulations on subgrids as detailed below.

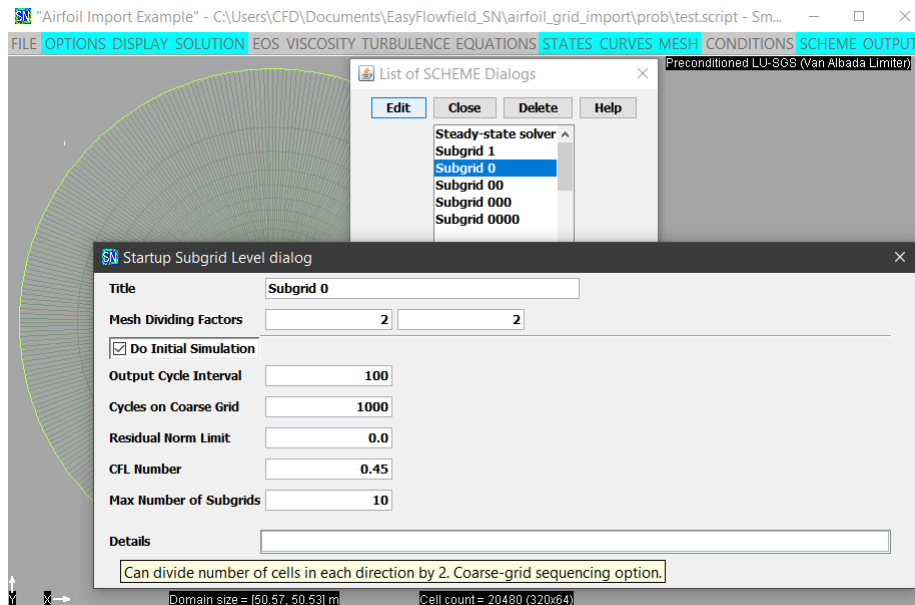


Fig. 5: Setting up an initial simulation on Subgrid 0.

First select "Subgrid 1" from the Startup Subgrid Level dialog list and click on Edit. Activate 'Do Initial Simulation' and set 'Output Cycle Interval' and 'Cycles on Coarse Grid' to 1000. Close the dialog and repeat the process for "Subgrid 0" but set 'Output Cycle Interval' to 100. Next use New to create a new Startup Subgrid Level dialog and change the title to "Subgrid 000" and close the dialog. Repeat the process to create one more subgrid level with the name "Subgrid 0000". Note that the names are not critical so you may use different ones more to your taste. When you are finished there should be 5 subgrid levels. The multigrid simulation on "Subgrid 0" will use the three subgrids below it. Next open the Steady State dialog under menu heading **SOLUTION** and set 'Output Cycle Interval' to 1000. Please close the dialog, save the script, and run the simulation.

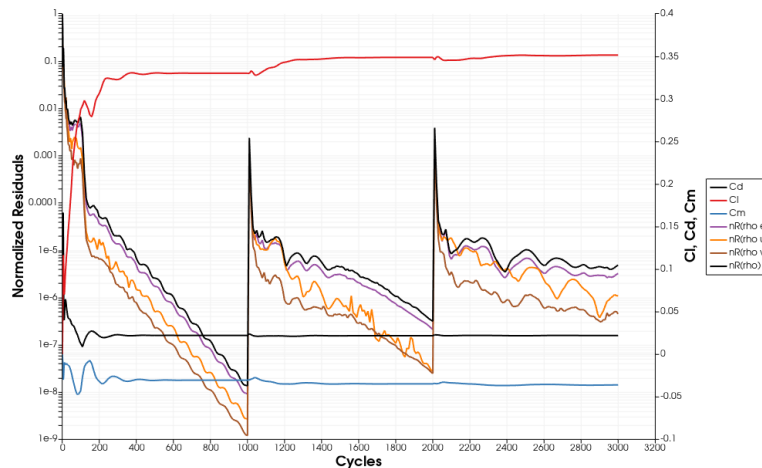


Fig. 6a: Convergence history with grid imported using CGNS format.

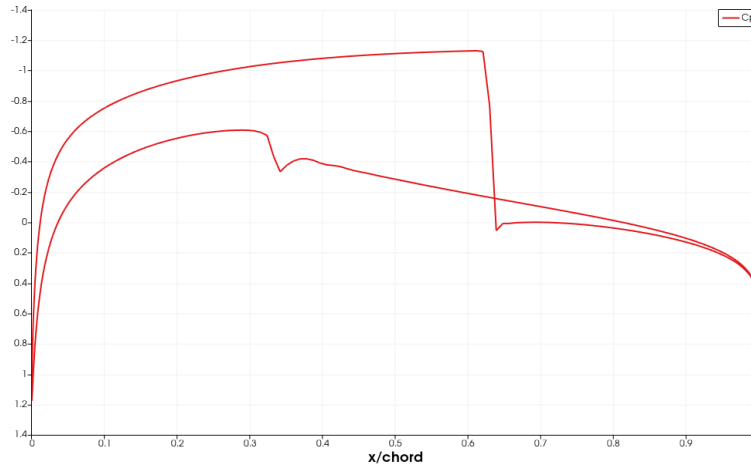


Fig. 6b: Cp along airfoil surface using LU-SGS solver on imported 320x64 cell grid (CGNS).

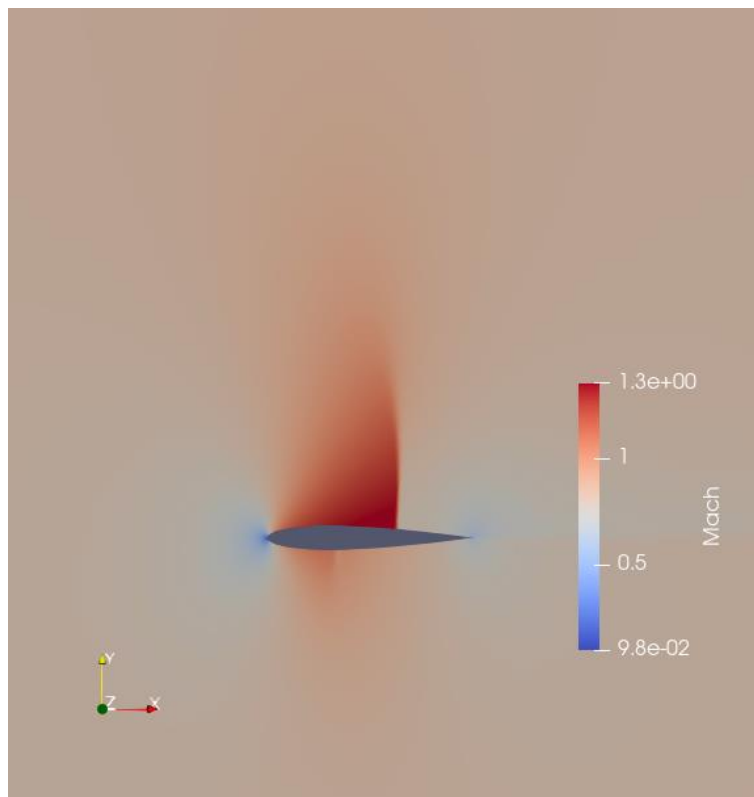


Fig. 6c: Mach contours using LU-SGS solver on imported 320x64 cell grid (CGNS).

Figures 6a through 6c display the resulting convergence history and solution. The convergence curves, force coefficients, Cp profiles, and Mach contours, are indistinguishable from those obtained in Tutorial 1. Please save test.script as test_CGNS.script so that you can rerun this version of the simulation at a future date.

We will now repeat the simulation using a grid saved in PLOT3D format. The main difference is that PLOT3D files do not store the boundary conditions or interface information. Please open test.script, open the Import CGNS Mesh dialog, and delete it.

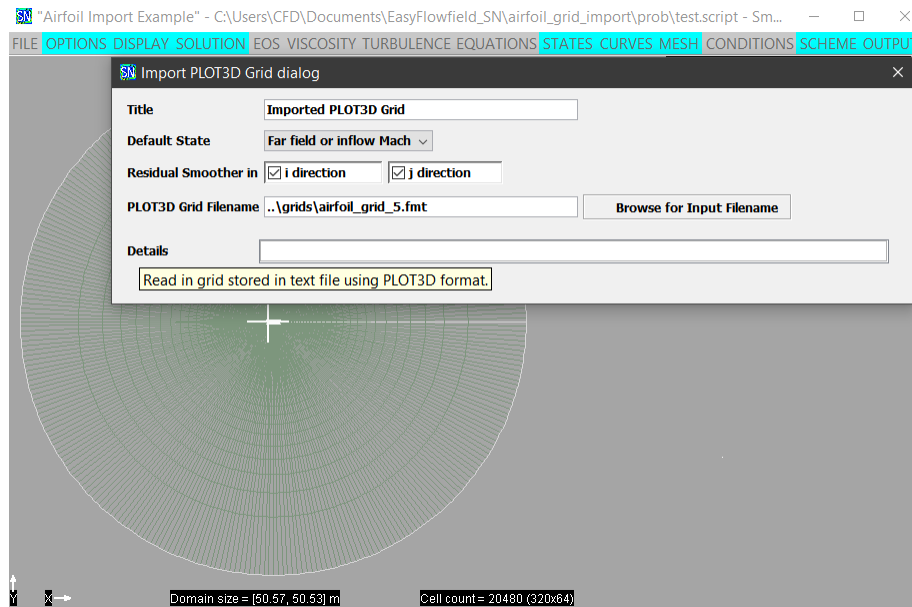


Fig. 7: Dialog used to import grid file in PLOT3D format.

Next open the Import PLOT3D Grid dialog, click on Browse for Input Filename and select "airfoil_grid_5.fmt". The grid will appear after the dialog is closed. Since boundary conditions are not stored in a PLOT3D grid file, all boundary conditions on the edges of the imported grid will be set to the Pass condition (white line).

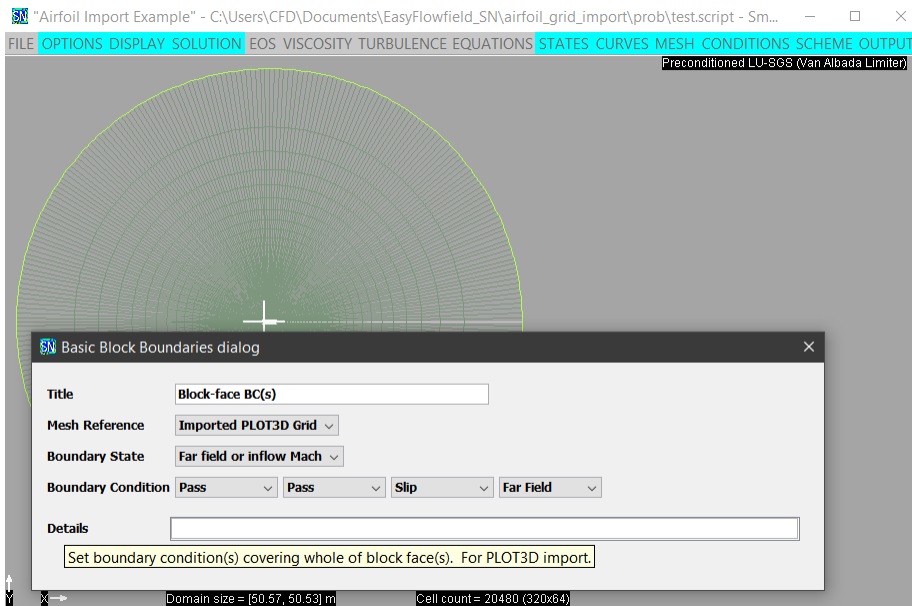


Fig. 8: Setting boundary conditions on edges of imported grid.

In order to specify the correct boundary conditions, open a Basic Block Boundaries dialog under Menu heading **CONDITIONS**. The first two boundary conditions (west and east) control conditions at the wake-line and must be set to Pass. The third boundary condition (south) controls conditions on the airfoil surface and is set to Slip. The fourth boundary condition (north) controls conditions on the outer edge of the o-grid and is set to Far Field (yellow). The new boundary colours will appear after you close the dialog.

Multiple instances of this dialog can be used to impose boundary conditions on each block in an imported multiblock grid. It can be used to override a boundary condition imposed by a CGNS file. Thus, a Slip condition on an airfoil or wing can be changed to a No Slip condition or vice versa.

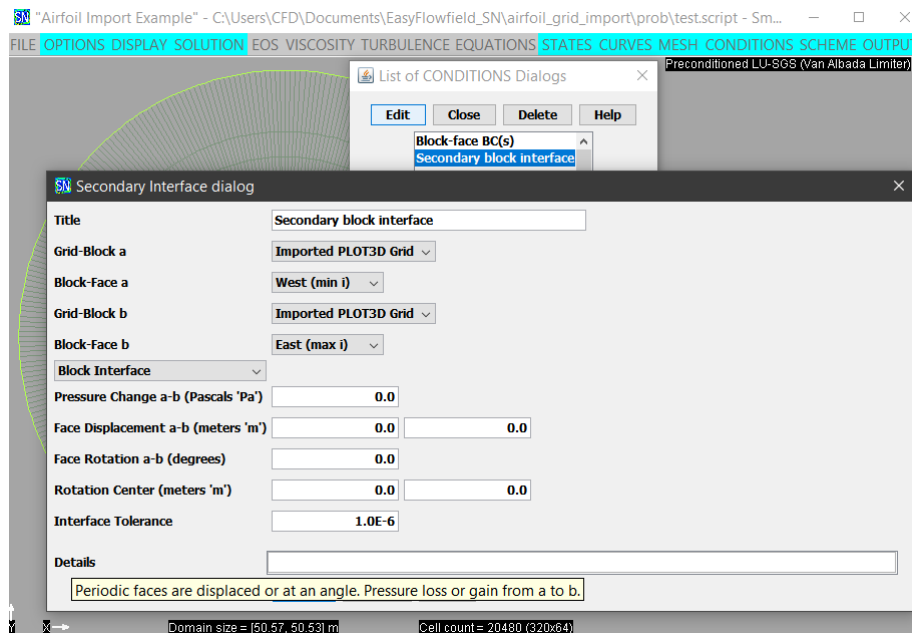


Fig. 9: Setting interface between adjacent faces of the o-grid.

Next open a Secondary Interface dialog and change the 'Periodic-Flow Boundary' option to the 'Block Interface' option. All other parameters can be left at their default values. This imposes an interface between the west and east faces of the o-grid which in this case meet along the wake line.

Multiple instances of this dialog can be used to join the faces of adjacent blocks in a multiblock grid. The 'Periodic-Flow Boundary' option can be used to join opposite faces of a grid.

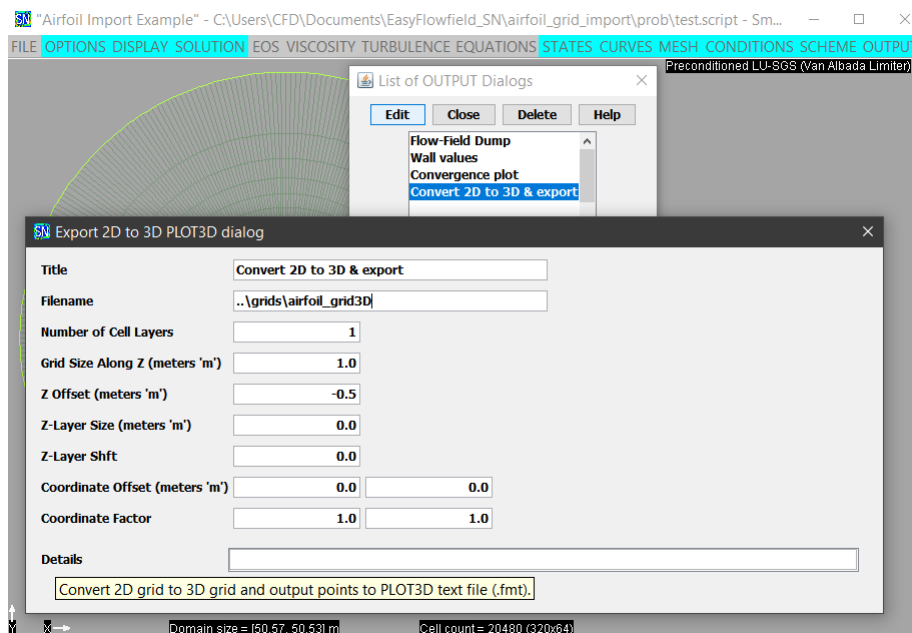


Fig. 10: Dialog used to convert the 2D grid to 3D and export it using the PLOT3D format.

The next and final example will demonstrate how to handle an imported 3D grid. In order to create this grid, please open the Export 2D to 3D PLOT3D dialog under menu heading **OUTPUT** and specify the filename "airfoil_grid3d". Please close the dialog, save the script, and run the simulation. A 3D grid based on the input 3D grid will be stored as airfoil_grid3D.fmt in airfoil_grid_import/grids at the start of execution.

The convergence curves, force coefficients, Cp profiles, and Mach contours, are indistinguishable from those obtained using the CGNS grid. Please save test.script as test_PLOT3D.script so that you can rerun this version of the simulation at a future date.

Please open test.script, open the Export 2D to 3D PLOT3D dialog and delete it. Next open the Steady State dialog under menu heading **SOLUTION** and change the number of dimensions to 3. You will get an error message when you close the dialog because the GUI expects the number of dimensions in the imported grid file to match the number of dimensions specified in the dialogs under menu heading **SOLUTION**.

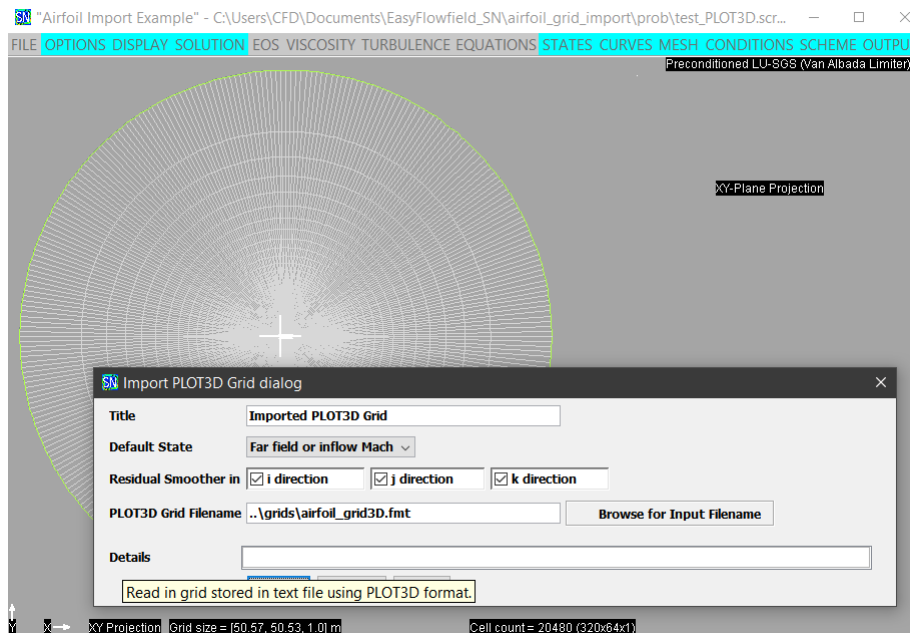


Fig. 11: Importing 3D grid in PLOT3D format.

To fix this, open the Import PLOT3D Grid dialog, click on Browse for Input Filename and select "airfoil_grid3D.fmt". The new grid will appear after the dialog is closed. The 3D grid has two new faces which are set to the default boundary condition of Pass (white).

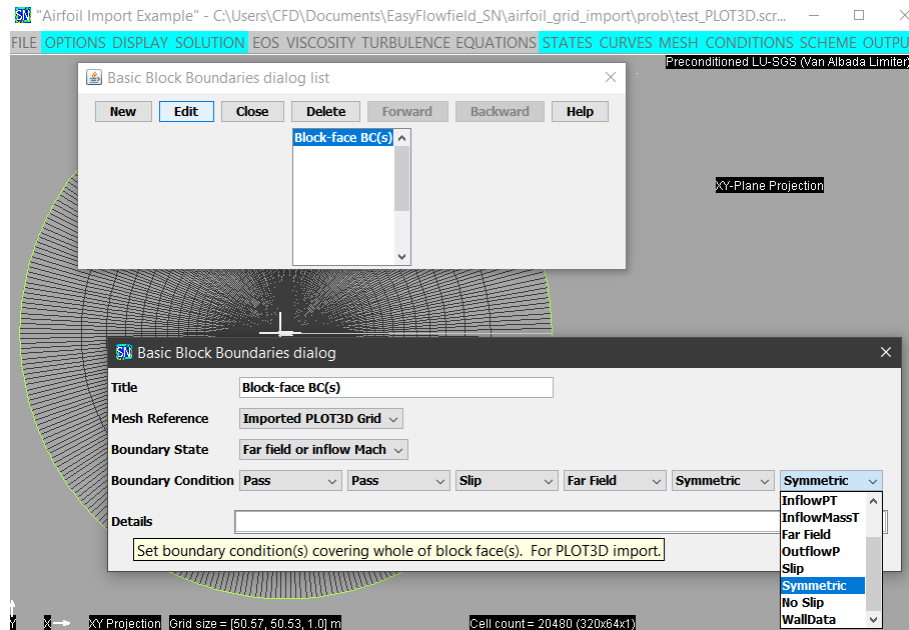


Fig. 12: Setting boundary conditions on 3D grid imported using PLOT3D format.

Please edit the Basic Block Boundaries dialog under menu heading **CONDITIONS** and set the last two boundary conditions to Symmetric. The Symmetric condition is treated like a Slip boundary condition, but no wall values or wall forces are output. The grid display will change after you close the dialog. Thin boundary lines selected in Flow Region Display dialog under menu heading **DISPLAY** show the grid lines most clearly in the above figure. After closing the dialog try holding down the Ctrl key and clicking on the edges and interior of the grid. Labels will appear summarizing the boundary conditions at the east, south, north, lower, and upper faces of the grid. You may have to zoom in or change your projection-plane option in the Flow Region Display dialog to see some of the labels. Note that in this case, the label for the west boundary condition is overridden by the label for the east boundary condition.

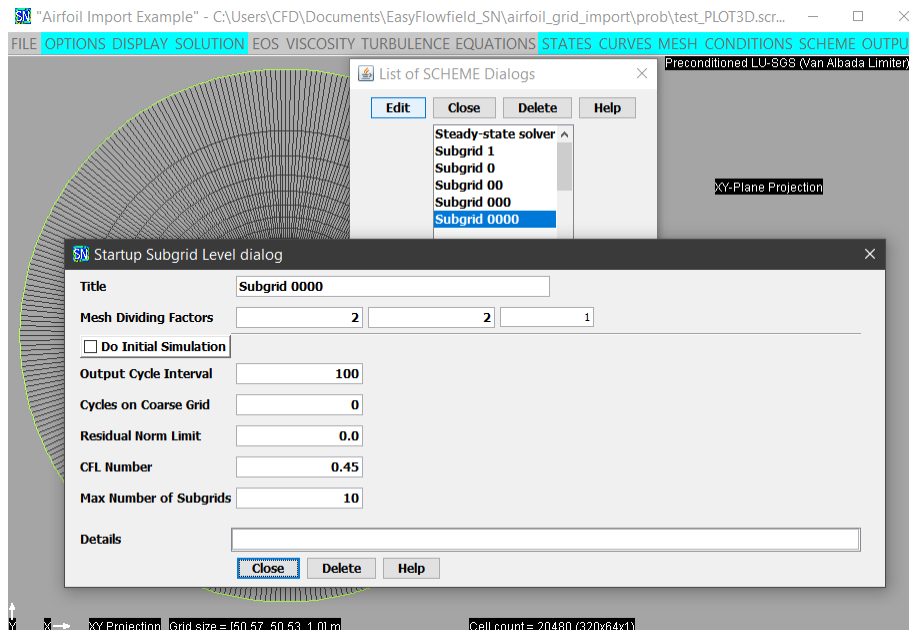


Fig. 13: Setting final mesh dividing factor to 1 in all subgrids.

Next, please edit all the Subgrid Level dialogs under menu heading **SCHEME**. Set the final mesh dividing factor to 1 since the grid is only one cell thick in the z direction.

You should also open the Steady-State Solver dialog and disable application of fluxes in the k direction (i.e. the z direction). If a 3D grid is only one cell thick in a certain direction, the size of the time step in some cells might be increased by disabling of the application of fluxes in that direction.

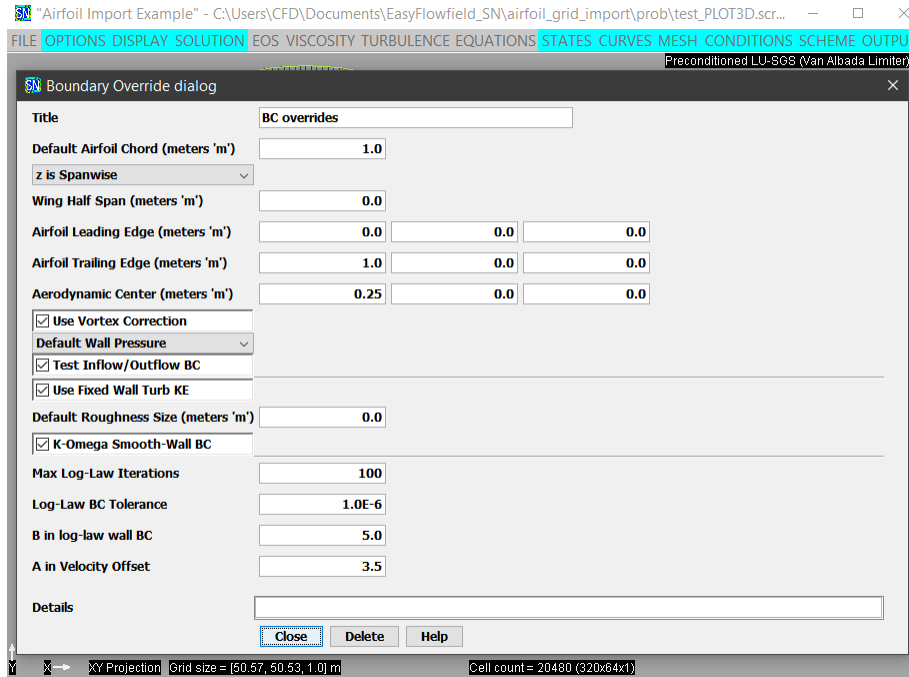


Fig. 14: Resetting wall pressure option.

Next open the Boundary Override dialog under menu heading **OPTIONS** and replace the 'Curved-Wall Pressure' option by the 'Default Wall Pressure' option. The 'Curved-Wall Pressure' option is not available for a 3D grid. This change prevents the solver from producing warning messages. You could also set the wing half-span to a nonzero value which would modify the 2D vortex correction into a 3D horseshoe-vortex correction but that is inappropriate in this case.

Please close the dialog, save the script, and run the simulation. The results are displayed in Figures 15a through 15c. In this case, the force coefficients and C_p curves differ noticeably from those obtained from the 2D simulations. This is caused by the absence of the 'Curved-Wall Pressure' option. This can be proved by rerunning test_PLOT3D.script with the 'Curved-Wall Pressure' option unselected. This difference is only significant with the relatively coarse grids used for inviscid simulations. Please save the file test.script as test_PLOT3D_3D.script so that you can rerun the simulation at a future date.

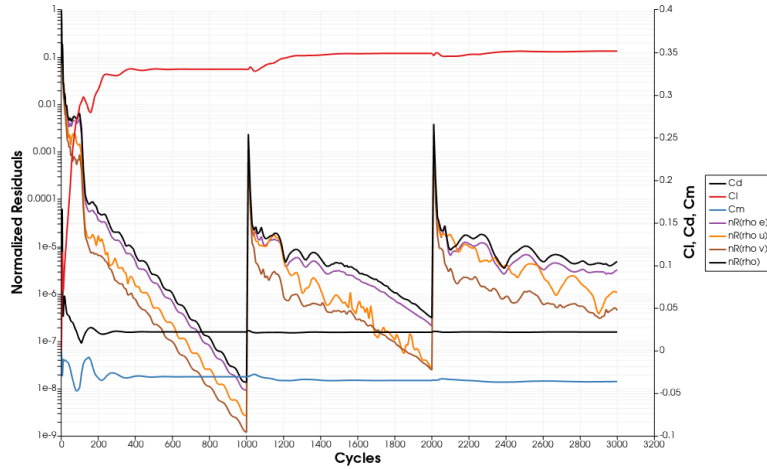


Fig. 6a: Convergence history using imported 320x64x1 cell grid (PLOT3D).

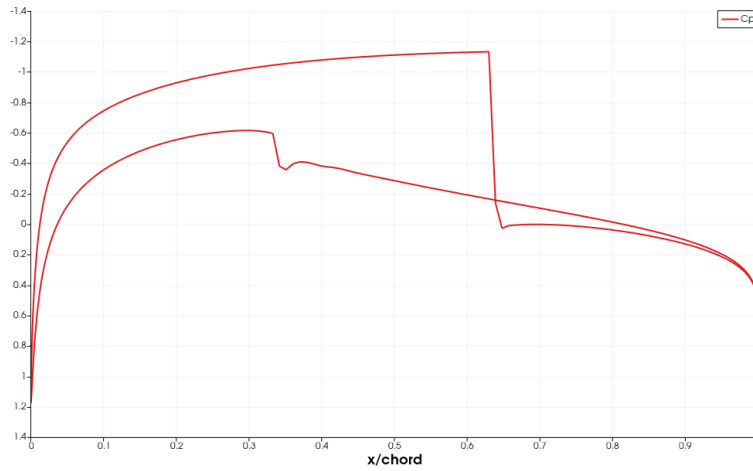


Fig. 6b: C_p along airfoil surface using LU-SGS solver on imported 320x64x1 cell grid (PLOT3D).

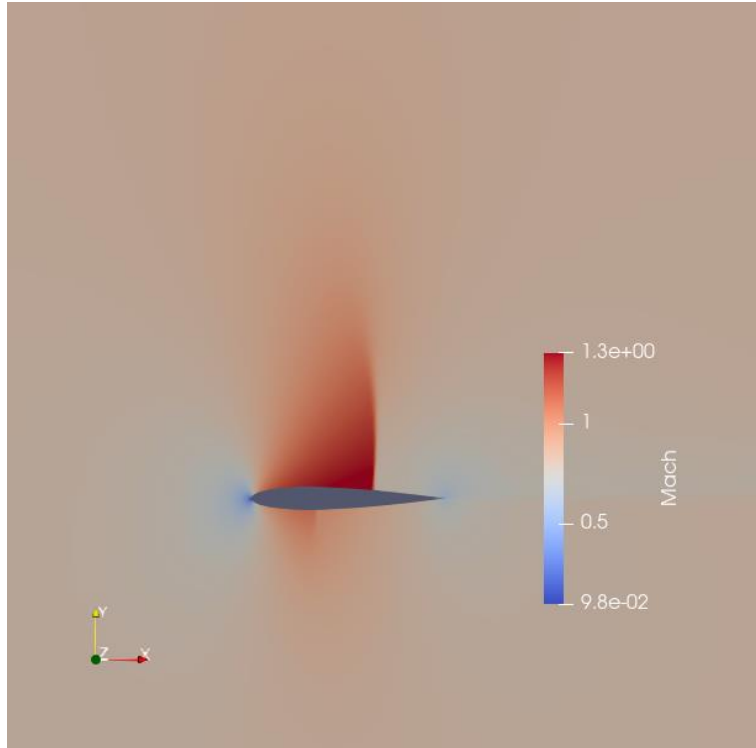


Fig. 6c: Mach contours using LU-SGS solver on imported 320x64x1 cell grid (PLOT3D).