EasyFlowfield Tutorial 3: Simulating Steady-State Flow Through a Nozzle
SmartNumerics Simulation Solutions Inc.

Version 6
June 4, 2020

This tutorial involving inviscid flow, covers application of inflow and outflow boundary conditions, creation of a grid with a curved boundary, and use of convergence acceleration options. ParaView will be used to produce contour plots and plots of pressure and Mach number along the axis of the nozzle.

This tutorial is based on a verification test case provided by the NPARC Alliance and is posted on the web (http://www.grc.nasa.gov/WWW/wind/valid/cdv/cdv.html). This page contains a link to test results produced by the Wind CFD code. The nozzle cross-sectional area has the form

$$A = \begin{cases} 1.75 - 0.75\cos((0.2x-1)\pi) & \text{for} \quad 0 \le x \le 5 \\ 1.25 - 0.25\cos((0.2x-1)\pi) & \text{for} \quad 5 \le x \le 10 \end{cases}$$

The plenum (reservoir) total pressure and temperature are 1 PSI and 100°R, respectively. Numerical results and analytical solutions are provided for exit static pressures of 0.89, 0.75, and 0.16 PSI, which correspond to subsonic outflow with isentropic flow, subsonic outflow with normal shock in the diffusing section, and supersonic outflow with isentropic flow, respectively. All three conditions are covered in detail in one of the EasyFlowfield validation documents.

You should first open the Unit Selection dialog under menu heading **OPTIONS** and set the pressure unit to PSI, the temperature unit to degrees Rankine, and the distance unit to inches. Next open the Simulation Overview dialog and change the title to "Nozzle".

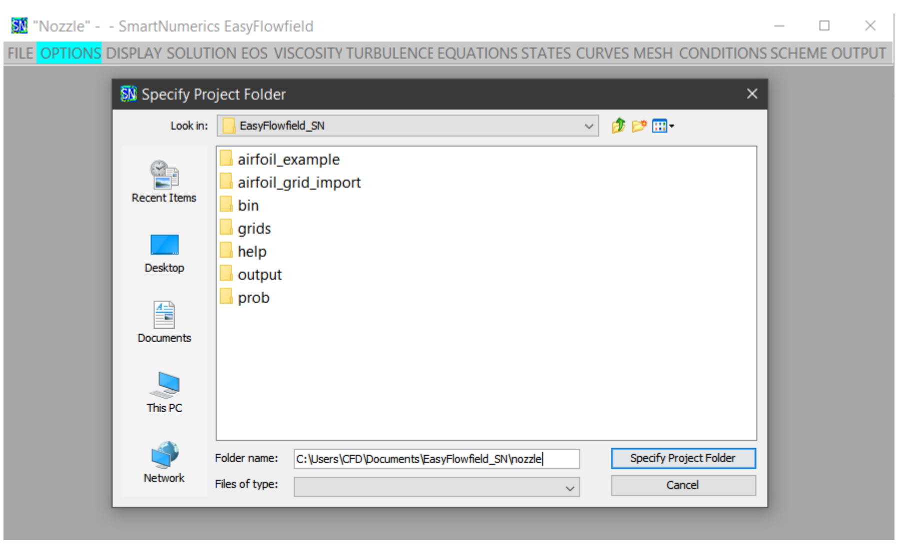

Fig. 1: Create project folder for nozzle simulation.

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

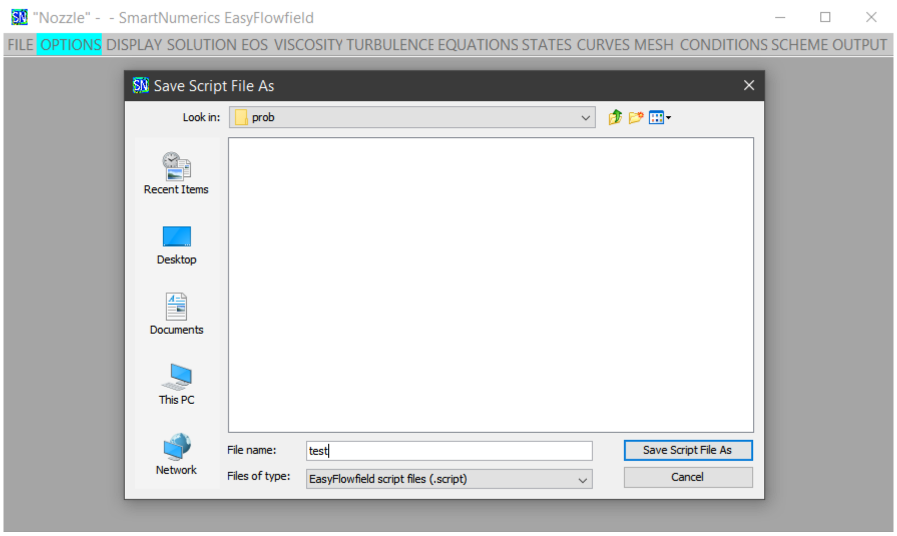

Fig 2: Save test.script in nozzle/prob.

Note that the nozzle folder has the subfolders, grids, output, and prob. To save the script, click on Save Script under menu heading **FILE**, navigate to nozzle/prob, enter the filename "test", and click on 'Save Script File As'.

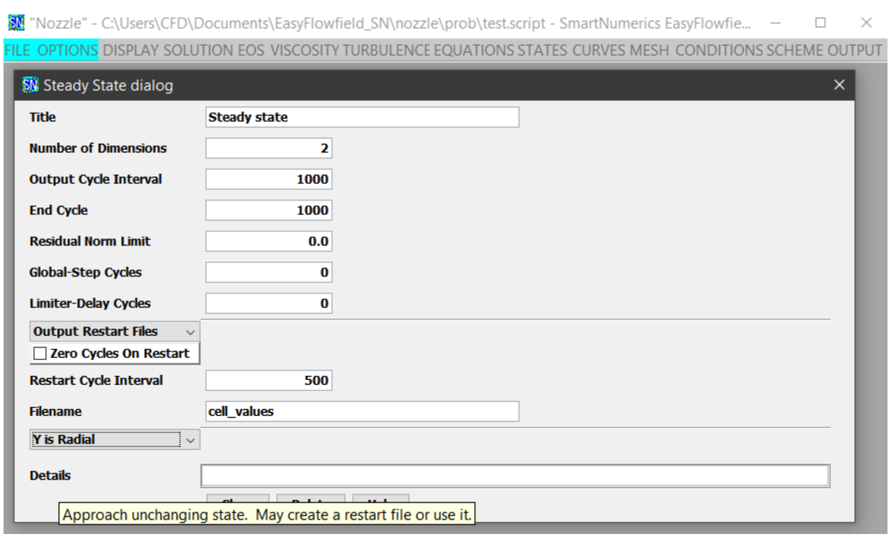

Fig. 3: Specify steady-state simulation with axial symmetry.

Then open the Steady State dialog under menu heading **SOLUTION** and set ‘Output Cycle Interval’ and ‘End Cycle’ to 1000. Also select output of restart files with an interval of 500 cycles and specify that the y-axis has cylindrical symmetry (radial).

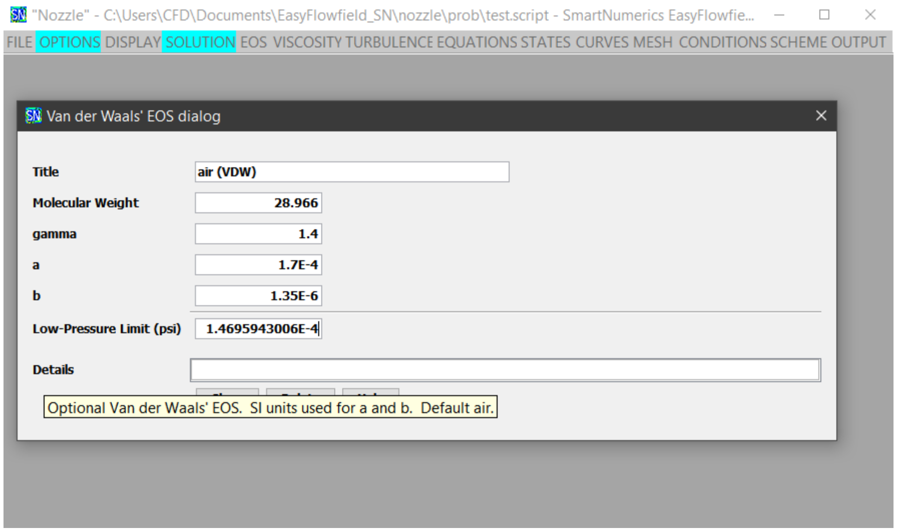

Fig. 4: Create EOS dialog and reduce low pressure limit by a factor of 100.

Next, click on Van der Waals' EOS under menu heading **EOS** and reduce the low-pressure limit by a factor of 100. Also, open and close the Euler dialog under menu heading **EQUATIONS** without making any changes. The checkbox in the Euler dialog enables the dialogs under menu heading **STATES** to list the van der Waals' EOS. In this case, use of an ideal gas EOS dialog would give similar results.

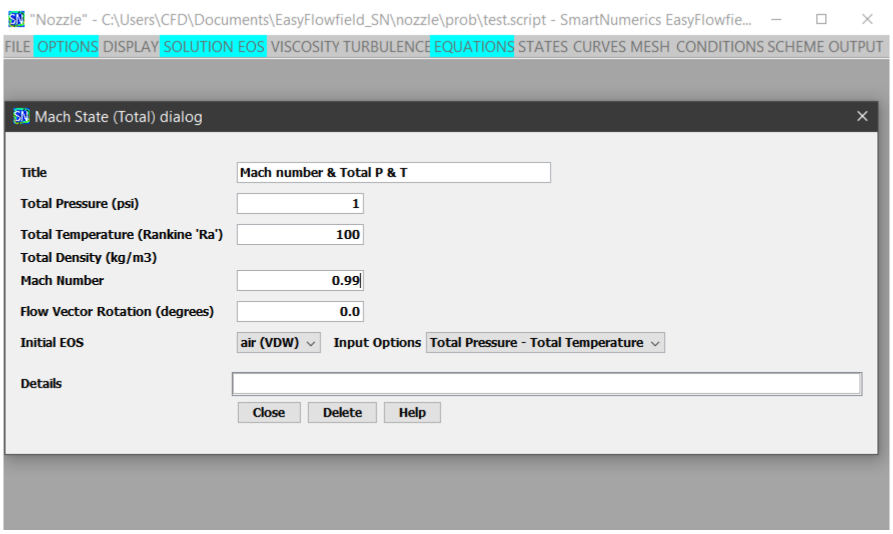

Fig. 5: Specify total pressure, total temperature, and Mach number.

Then click on Mach Number (Total) under menu heading **STATES**. Specify a total pressure of 1 pound per square inch, a total temperature of 100° Rankine, and an initial Mach number of 0.99.

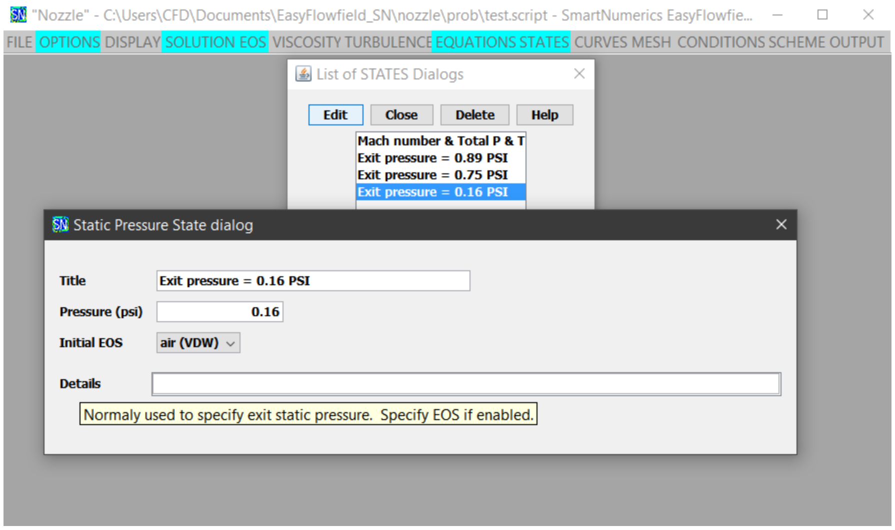

Fig. 6: Create outflow pressure states.

Next click on Static Pressure State under menu heading **STATES** and set the pressure to 0.89 PSI and change the title to "Exit Pressure = 0.89". Close the dialog and use New to create two more dialogs with the pressure set to 0.75 PSI and 0.16 PSI.

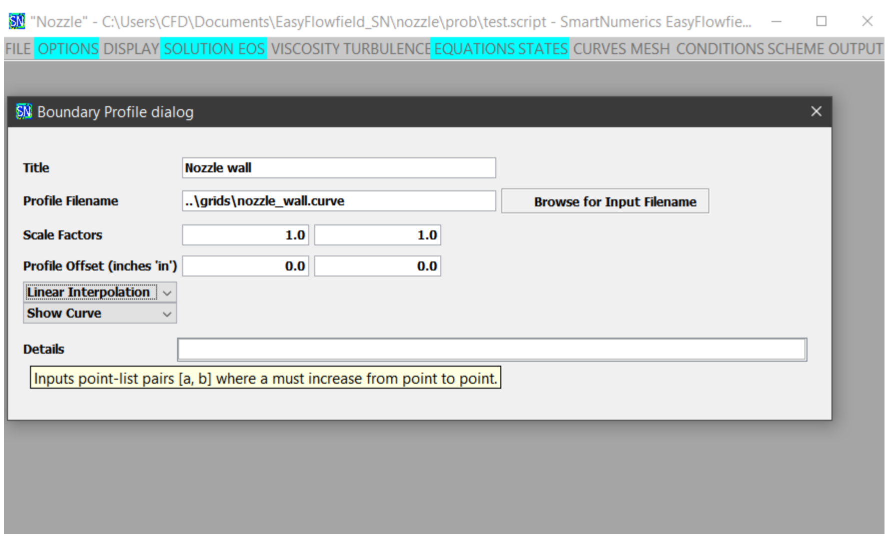

Fig. 7: Specify boundary profile.

The nozzle has a curved boundary defined by the file "nozzle_wall.curve" that contains a list of xy points that can generated by a spread sheet application such as Microsoft Excel using

$$y = \begin{cases} \sqrt{\dfrac{1.75 - 0.75\cos((0.2x-1)\pi)}{\pi}} & \text{for} \quad 0 \le x \le 5 \\ \sqrt{\dfrac{1.25 - 0.25\cos((0.2x-1)\pi)}{\pi}} & \text{for} \quad 5 \le x \le 10 \end{cases}$$

Please copy this file from EasyFlowfield_SN/grids to nozzle/grids. Then open a Boundary Profile dialog under menu heading **CURVES**, click on Browse for Input Filename, highlight the curve file, and click on Select Profile Filename. Then select 'Linear Interpolation' and change the title to "Nozzle wall".

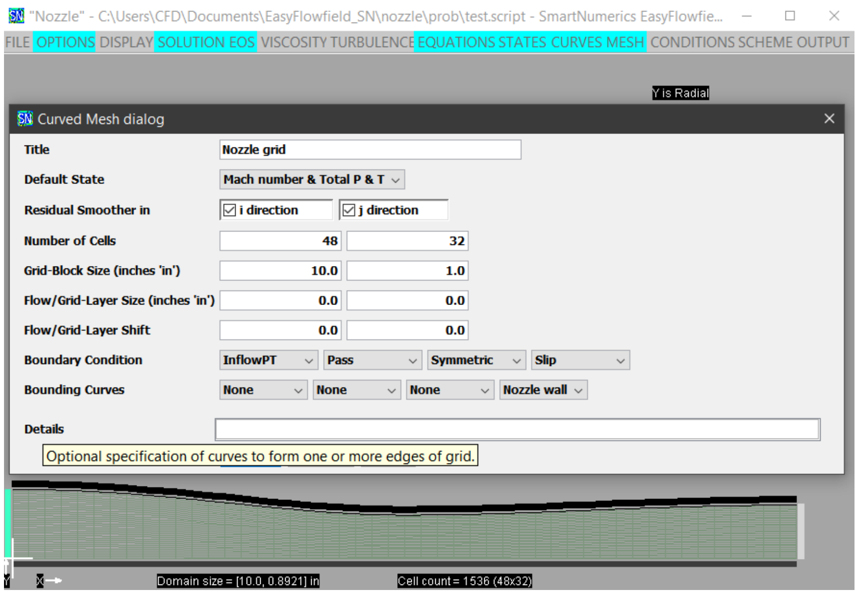

Fig. 8: Specify parameters and boundary conditions for grid.

Next open the Curved Mesh dialog under menu heading **MESH** and specify a 48x32 cell grid with dimensions set to 10 inches by 1 inch. The boundary conditions should be set to InflowPT, Pass, Symmetric, and Slip. The InflowPT condition imposes the 'Default State' values of total pressure and total temperature at the inlet (west). The Slip and Symmetric conditions impose refection conditions at the north and south boundaries. Please set the north boundary curve to "Nozzle wall" and the title to "Nozzle grid".

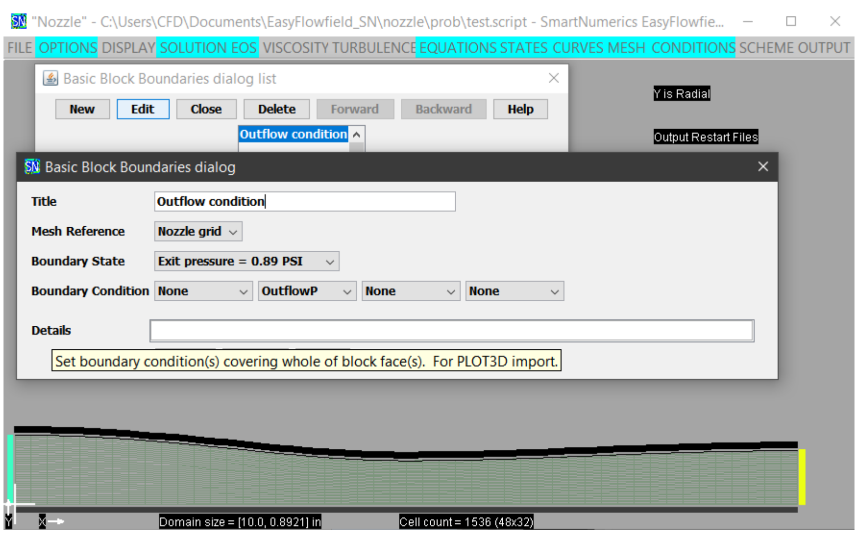

Fig. 9: Specify outflow boundary condition.

Next click on a Basic Block Boundaries under menu heading **CONDITIONS** and select the exit-pressure option of 0.89 PSI for 'Boundary State'. Set the east boundary condition to OutflowP which imposes the specifed value of static pressure at the outlet. Please change the title to "Outflow condition". The secondary-boundary dialogs available under menu heading **CONDITIONS** can also be used to set the outflow condition but are a bit more involved. To confirm the boundary conditions, close the dialog, press the Ctrl key on the keybord, and click on each edge of the grid. A label will appear with details.

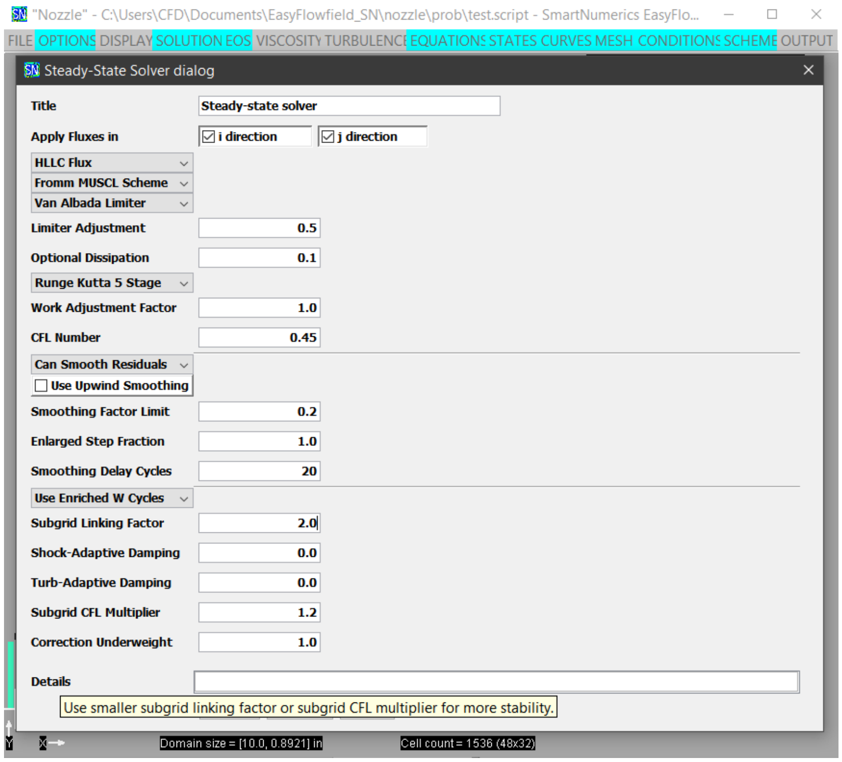

Fig. 10: Specify solver parameters.

Then open the Steady-State Solver dialog under menu heading **SCHEME**, set 'Optional Dissipation' to 0.1, select ‘Use Enriched W Cycles’, and select 'Can Smooth Residuals' to improve the rate of convergence to the steady state. Please leave all the other parameters at their default values.

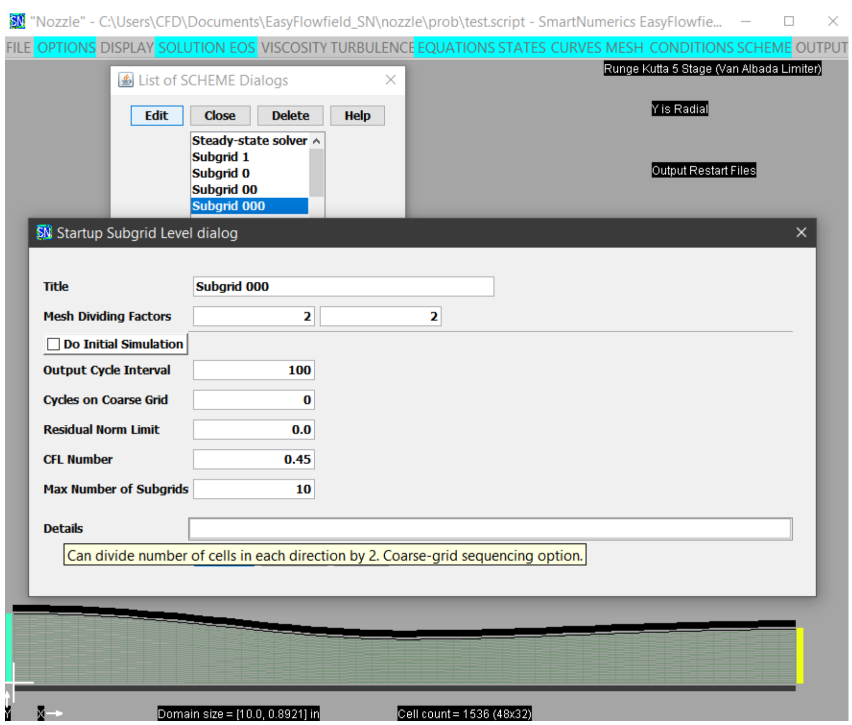

Fig. 11: Create subgrids.

Next open four Startup Subgrid Level dialogs under menu heading **SCHEME**. The titles of the subgrids are not critical as long as they differ. Each subgrid is obtained from a finer grid by dividing the number of cells in each direction by two.

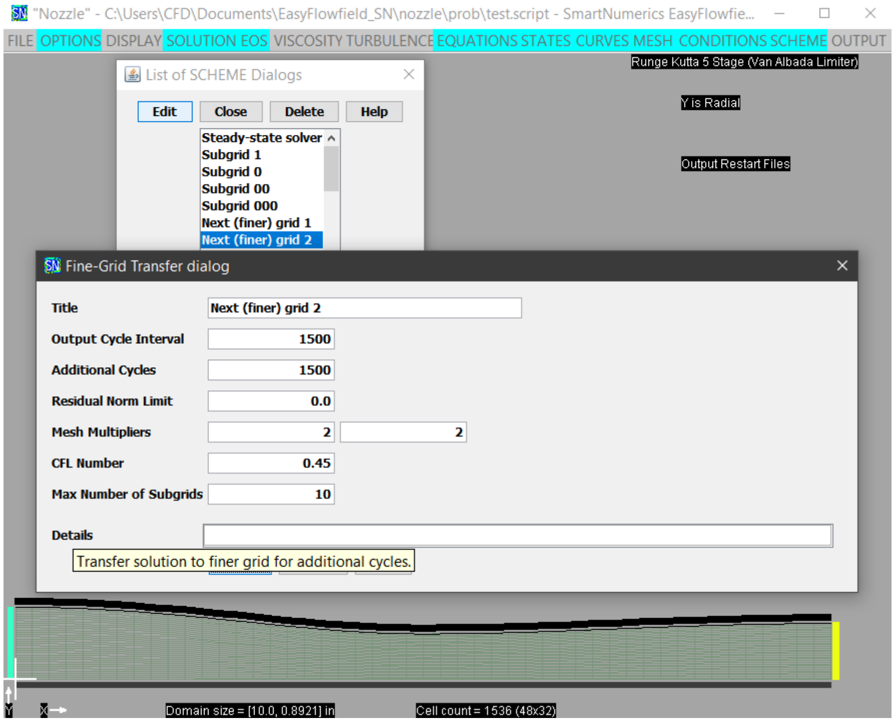

Fig. 12: Creating finer grids.

Then create two Fine-Grid Transfer dialogs under menu heading **SCHEME**. Please set 'Output Cycle Interval' and 'Additional Cycles' to 1000 for the first fine grid. Please set these parameters to 1500 cycles for the second fine grid. The first fine grid will be a 96x64 cell grid and the second will be a 192x128 cell grid.

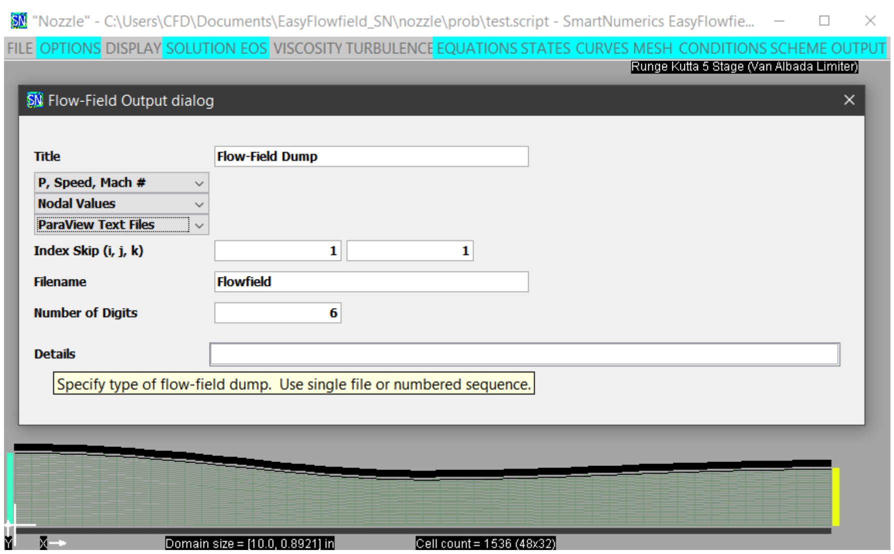

Fig. 13: Specify flow-field output.

Next open the Flow-Field Output dialog under menu heading **OUTPUT** and specify output of pressure, flow speed, and Mach number. Also specify output of values at the nodes (cell corners) and use of the ParaView Text File format.

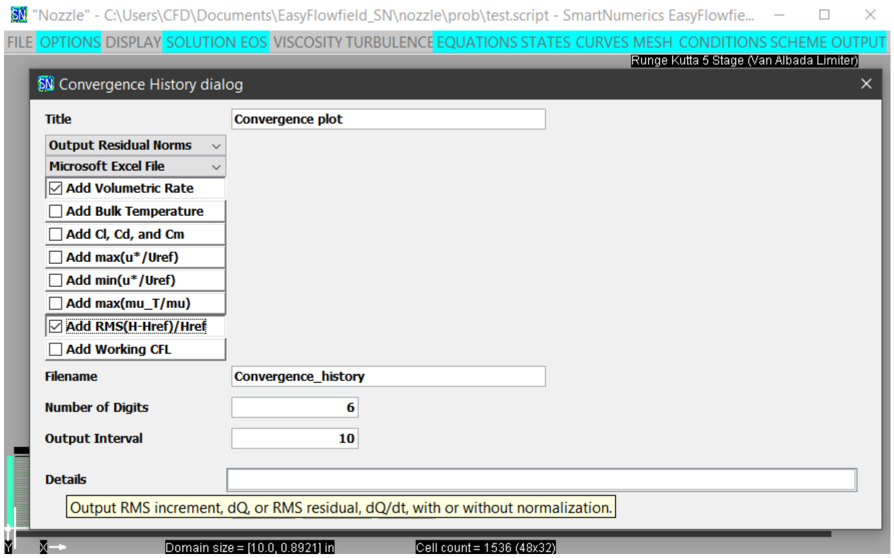

Fig. 14: Specify convergence monitoring output.

Please open the Convergence History dialog and specify Microsoft Excel file format. Select 'Add volumetric Rate' which will output the volumetric rate at the outflow boundary. Also select 'Add RMS(H-Href)/Href' which is a measure of the departure of the total enthalpy in the flow from the inflow total enthalpy. This should be zero for inviscid flow. Numerical dissipation will raise it slightly above zero with the greatest departure being on the coarsest grid.

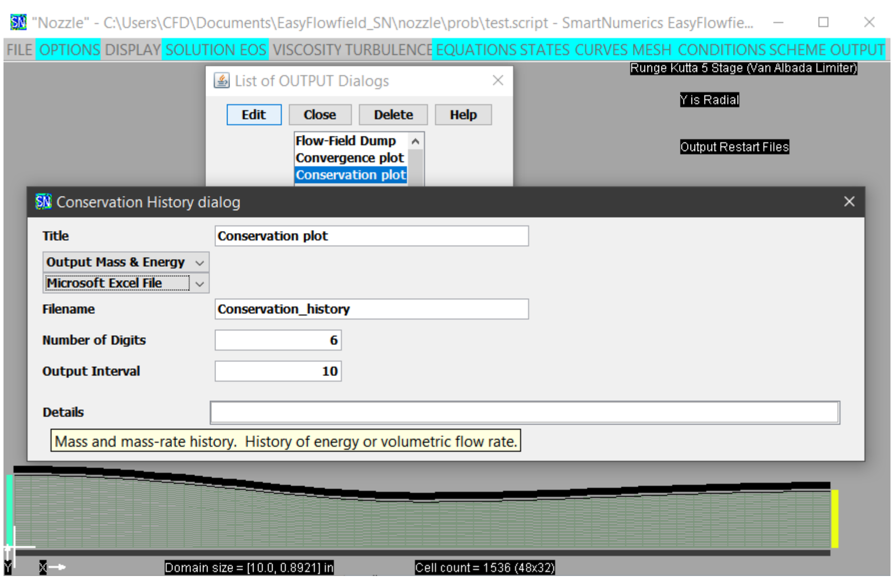

Fig. 15: Specify conservation output.

Next open the Conservation History dialog and specify output of the Microsoft Excel file format (.csv). Then, close the dialog, save the script, and run the simulation using the Automated Simulation option under menu heading **FILE**. While the simulation is running, please save test.script as test_p_89.script for future reference.

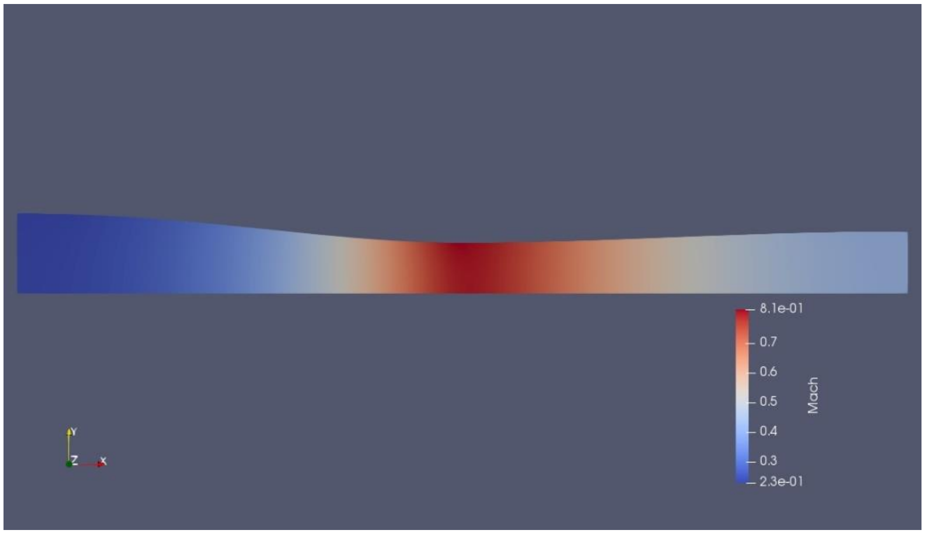

Fig. 16a: Mach contours on finest grid with exit pressure of 0.89 PSI.

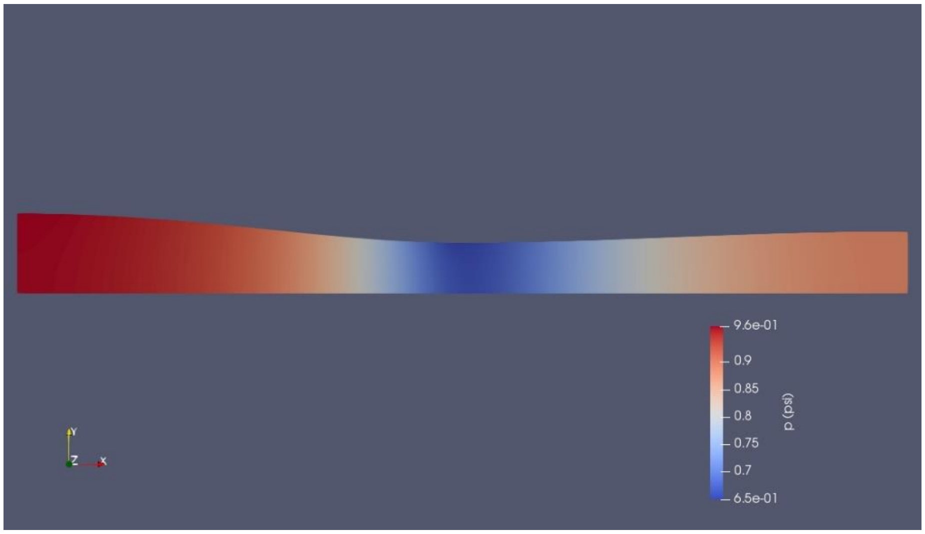

Fig. 16b: Pressure contours from finest grid with exit pressure of 0.89 PSI.

Figures 16a and 16b display Mach and pressure contours from the finest grid produced using ParaView. If desired, contour lines can be displayed by applying the ‘Contour’ filter in ParaView.

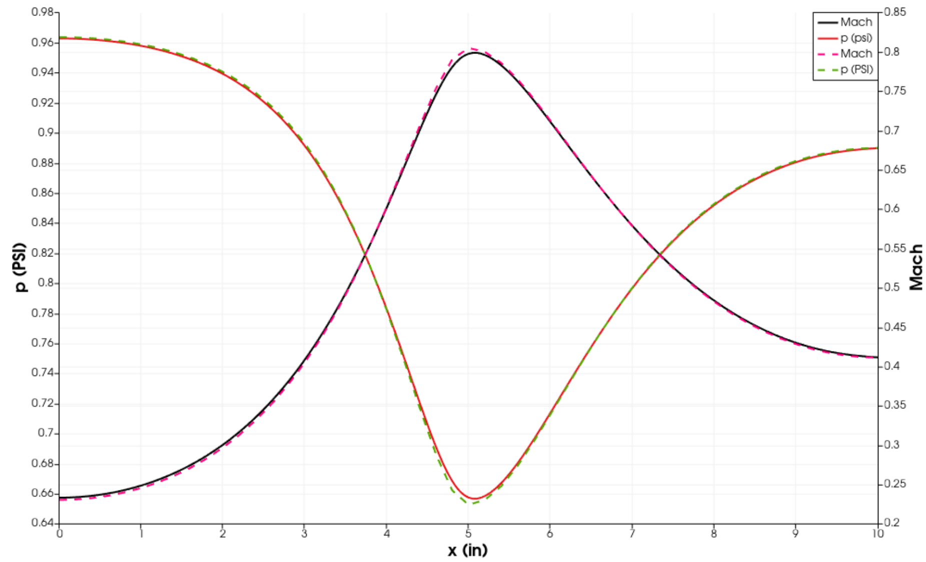

Fig. 16c: Pressure and Mach number from fine grid (solid) compared to reference curves (dashed).

Figure 16c compares the Mach number and pressure from the reference curves (dashed) for exit pressure set to 0.89 PSI to values along the axis of the fine grid. The curves were produced using ParaView as follows. The flow-field file (flowfield_5.vts) was input to ParaView and the 'ExtractSubset' filter was applied with the upper j index set to zero. The ParaView 'Save Data' command was used to save the center-line subset as a .csv file. The flow-field file was closed, and the center line .csv file and a text file written in .csv format containing the reference curves were then input to ParaView (input the reference file first). A 'PlotData' filter was applied to each file and appropriate line styles were applied to each curve. The Mach profile was assigned to the right axis by highlighting “Mach” in the list of variables and specifying ‘Bottom-Right’ instead of ‘Bottom-Left’. The reference text file was created by adding commas and removing comments from a text file originally written in Tecplot format. The filetype of the reference text file should be set to .txt since EasyFlowfield deletes all files with filetype .dat, .vts, or .csv when it initiates a simulation. Figure 16d below compares the Mach number and pressure from the reference curves for exit pressure set to 0.89 PSI to values along the axis of the coarse grid.

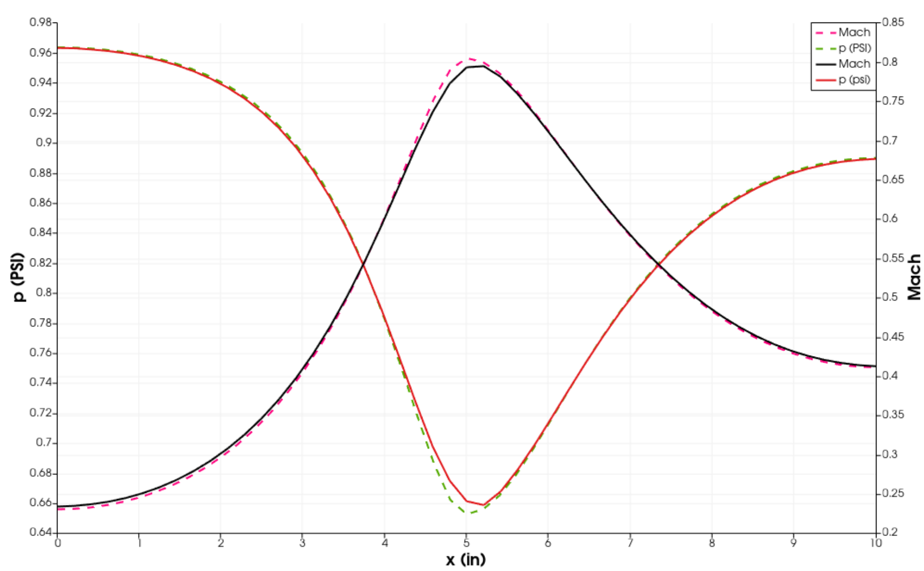

Fig. 16d: Pressure and Mach number from coarse grid (solid) compared to reference curves.

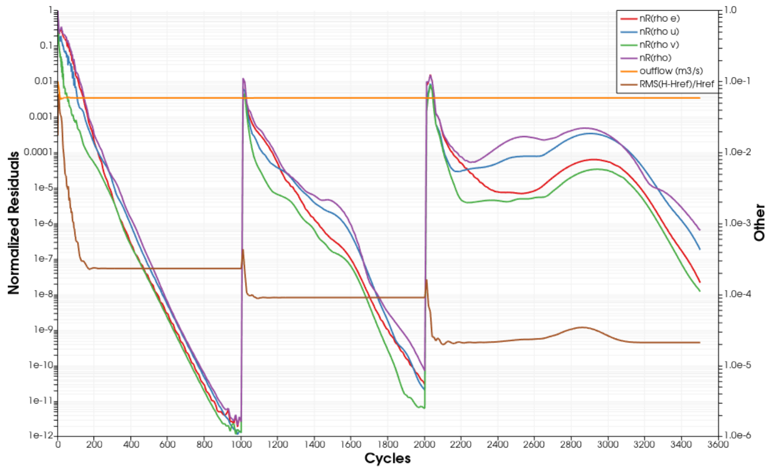

Fig. 16e: Convergence history with the exit pressure set to 0.89 PSI.

Figure 16e displays the convergence history produced using ParaView. The RMS values of all residuals decrease by almost five orders of magnitude as the simulation reaches a steady state on the finest grid. The measure of the uniformity of total enthalpy is smallest on the finest grid.

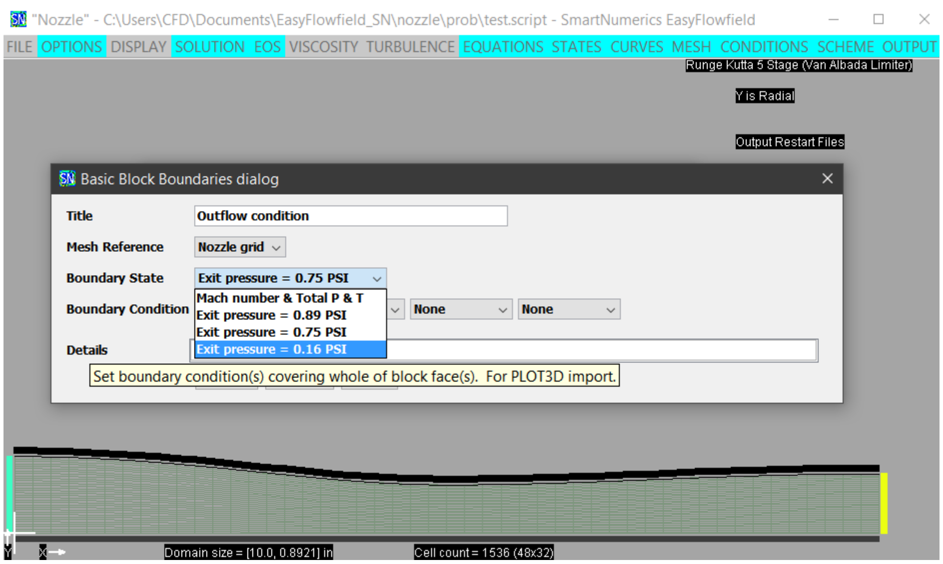

Fig. 17: Reduce exit pressure to 0.75 PSI.

To run the script with a different exit pressure, open "test.sript", open the Basic Block Boundaries dialog and set the exit pressure to 0.75 PSI or 0.16 PSI. Also open the Steady-State Solver dialog and reduce ‘Subgrid Linking Factor’ to 1. Figures 18a through 18e display the results with the exit pressure set to 0.75 PSI. All three conditions are covered in detail in one of the EasyFlowfield validation documents.

Please try plotting the convergence history as a function of work instead of cycles Please try re-running the simulations with ‘Use w Cycles’ selected in the Steady-State Solver dialog.

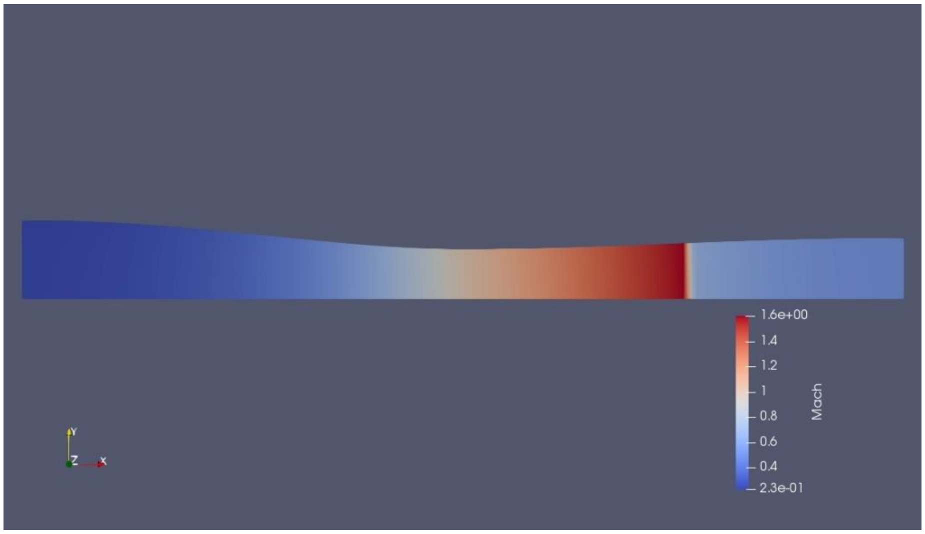

Fig. 18a: Mach contours on finest grid with exit pressure of 0.75 PSI.

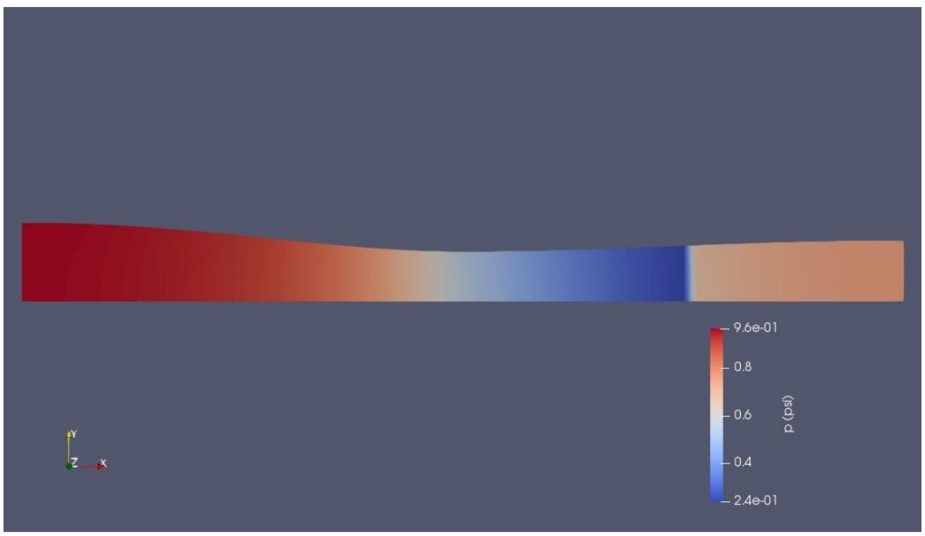

Fig. 18b: Mach contours on finest grid with exit pressure of 0.89 PSI.

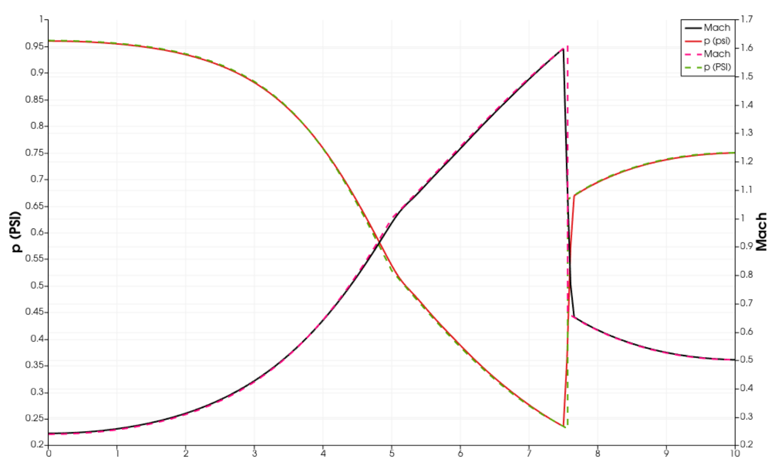

Fig. 18c: Pressure and Mach number from fine grid (solid) compared to reference curves.

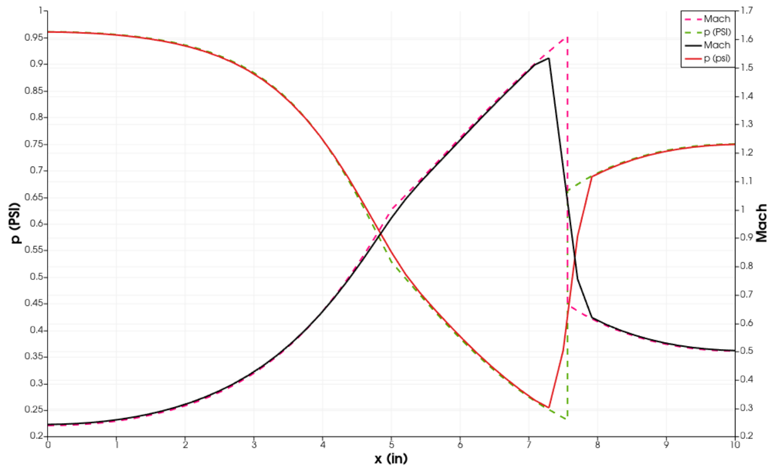

Fig. 18d: Pressure and Mach number from coarse grid (solid) compared to reference curves.

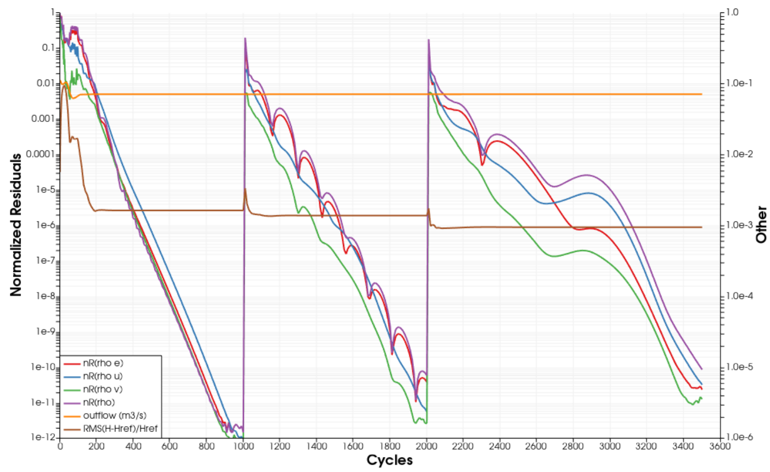

Fig. 18e: Convergence history with the exit pressure set to 0.75 PSI.

The RMS values of all residuals decrease by almost three orders of magnitude as the simulation reaches a steady state on the finest grid. The values are much larger than in the previous case due to the stationary shock front located downstream of the nozzle throat.