**EasyFlowfield Tutorial 5: Shock-Tube Simulation**
**SmartNumerics Simulation Solutions Inc.**

**Version 6**
**June 5, 2020**



A shock tube a device used to generate shockwaves in order to study their effects. A shock tube is a tube of rectangular or circular cross section in which a region of high pressure is separated from a region of lower pressure by a diaphragm. When the diaphragm is broken, a shockwave moves into the low-pressure region and rarefaction wave moves into the high-pressure region. The rarefaction wave cools the high-pressure gas and the shockwave heats the low-pressure gas. The “contact surface” or region of contact or between the cooled gas and the heated gas moves downstream. The simple one-dimensional shock-tube simulation performed in this section is a standard problem used to test a CFD code.

This short tutorial covers control of a time-accurate simulation, imposition of a flow region, and use of a fine initial grid. Monitoring of flow conditions near a specified point is demonstrated. You can learn about the effect of changes in the flux limiter and other solver parameters. Use of spherical or cylindrical symmetry is also discussed. As a minimum, please do tutorial 1 first, for hints on how to use ParaView to plot data from .csv files.

You should first open the Unit Selection dialog under menu heading **OPTIONS** and set the pressure unit to atmospheres and the time unit to milliseconds. Next open the Simulation Overview dialog and change the title to "Shock Tube".

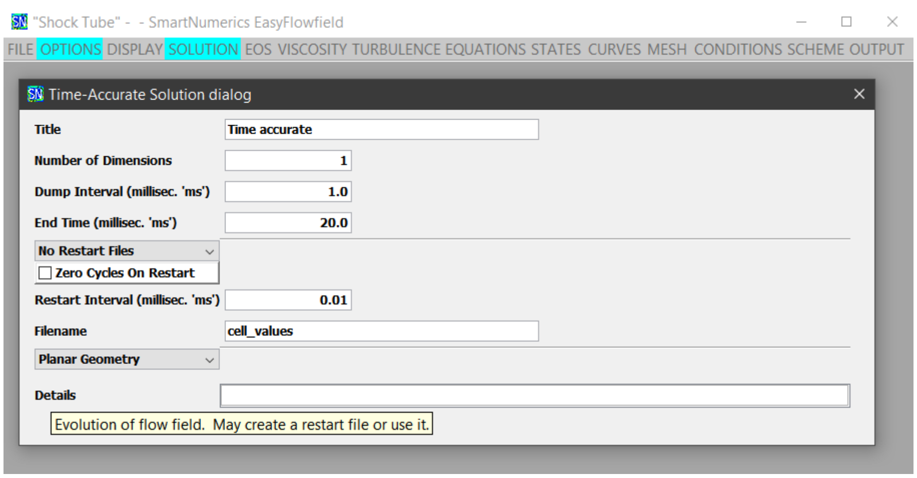


**Fig. 1: Dialog used to control simulation of transient flow.**

Next specify a transient flow solution by opening the Time-Accurate Solution dialog under menu heading **SOLUTION**. Specify a Dump interval of 1 millisecond, an end time of 20 milliseconds, and set the number of dimensions to 1. All other parameters should be left at their default values.

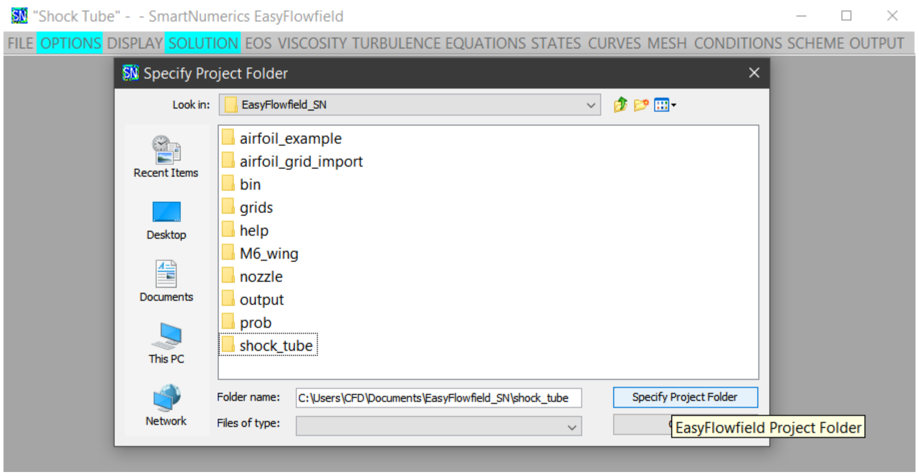


**Fig. 2: Create shock-tube project Folder.**

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

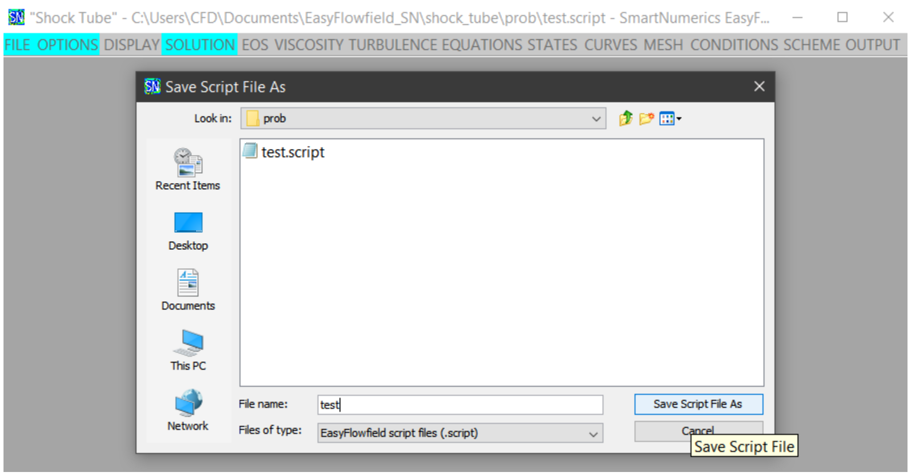


**Fig. 3: Saving new script file.**

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

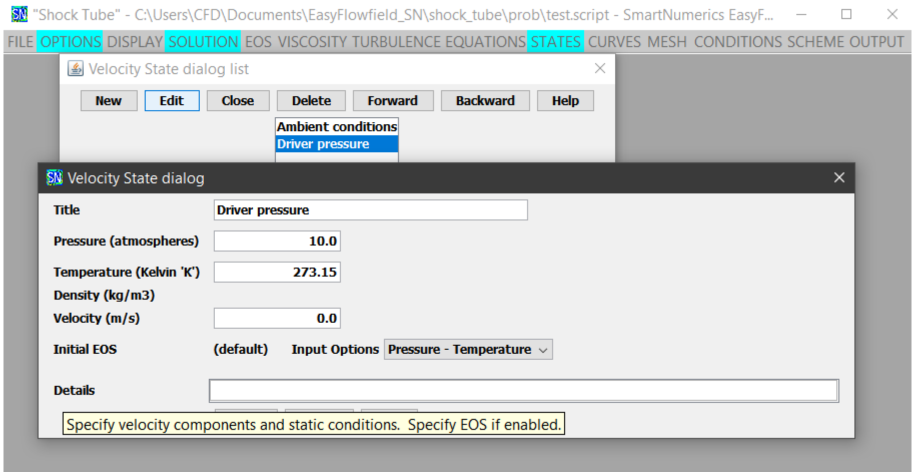


**Fig. 4: Defining initial flow states for the shock-tube problem.**

Next open and close a Velocity State dialog under menu heading **STATES** without changing any parameters. Then use New to create a second Velocity State, change the title to "Driver pressure", and specify a pressure of 10 atmospheres. The list of velocity state dialogs should be as depicted in Figure 4.

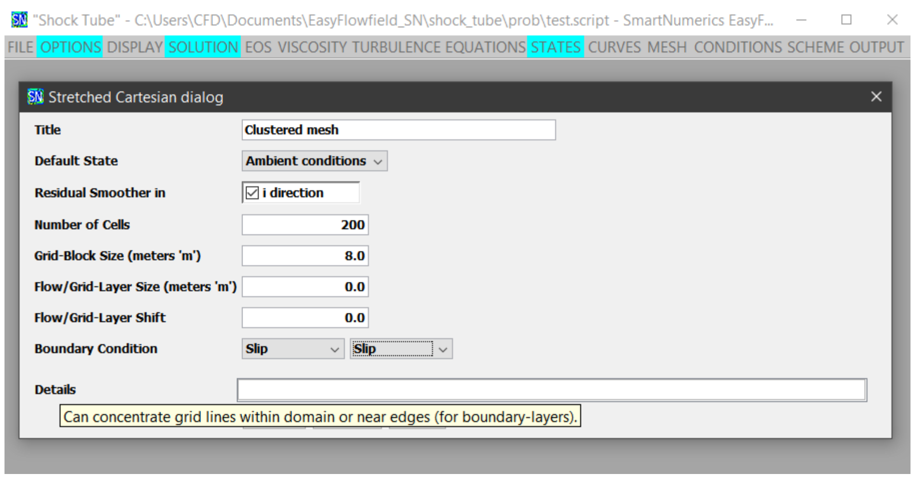


**Fig. 5: Grid for the shock-tube problem.**

Next open a Stretched Cartesian dialog under menu heading **MESH, s**et the number of cells to 200, and the grid-block size to 8 meters. Set the boundary condition at both ends of the grid to Slip (refection). The default state should already be set to ambient conditions. The grid will appear after you close the dialog.

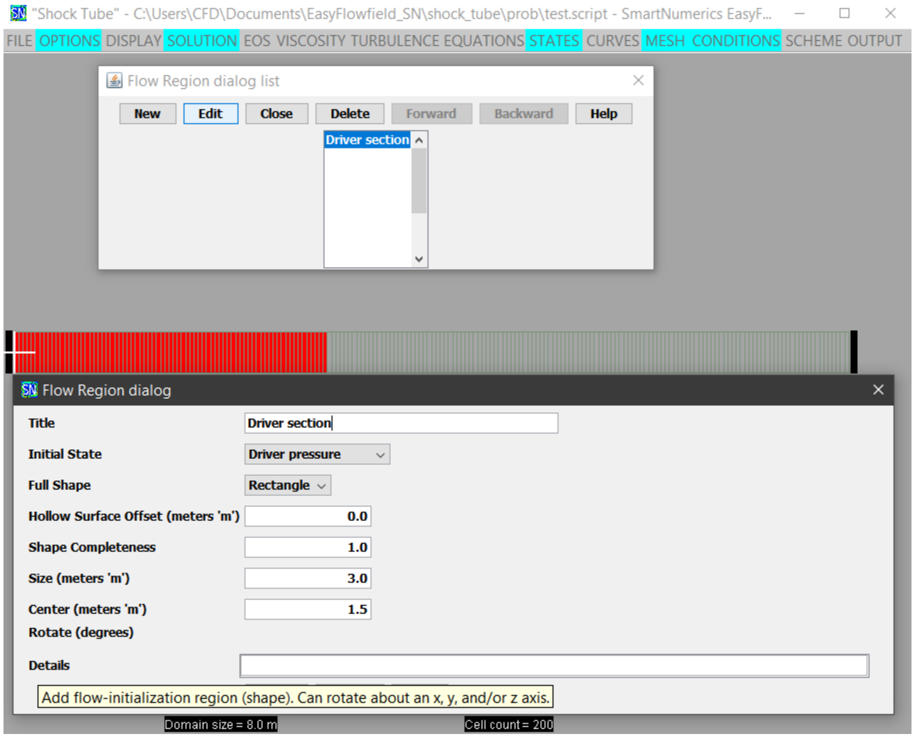


**Fig. 6: Defining driver flow region.**

Next open a Flow Region dialog under menu heading **CONDITIONS** and specify a rectangle three meters in length centered on 1.5 meters. Change the title to "Driver section". Set the initial state to Driver pressure. The driver region will appear after you close the dialog.

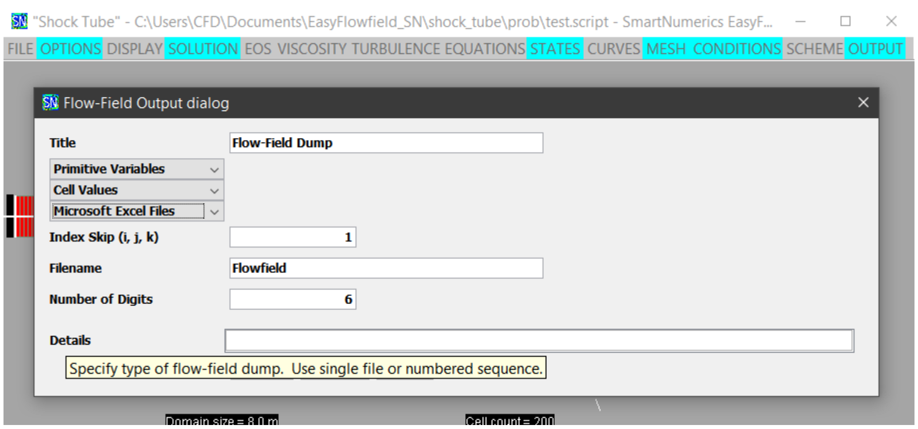


**Fig. 7: Specification of flow-field variables and output format.**

Next open the Flow-Field Output dialog under menu heading **OUTPUT** and specify output to Microsoft Excel Files. The flow values will be stored in the file Flowfield.csv written in "comma separated values" format.

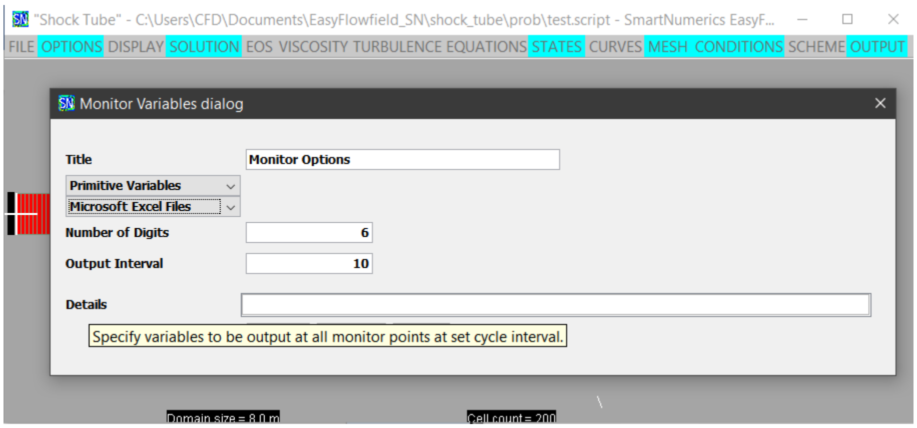


**Fig. 8: Specification of monitor variables and output format.**

Next open the Monitor Variables dialog. This dialog is used to specify the flow variables and file format used by monitor points with locations given as detailed below. Select the Microsoft Excel Files format.

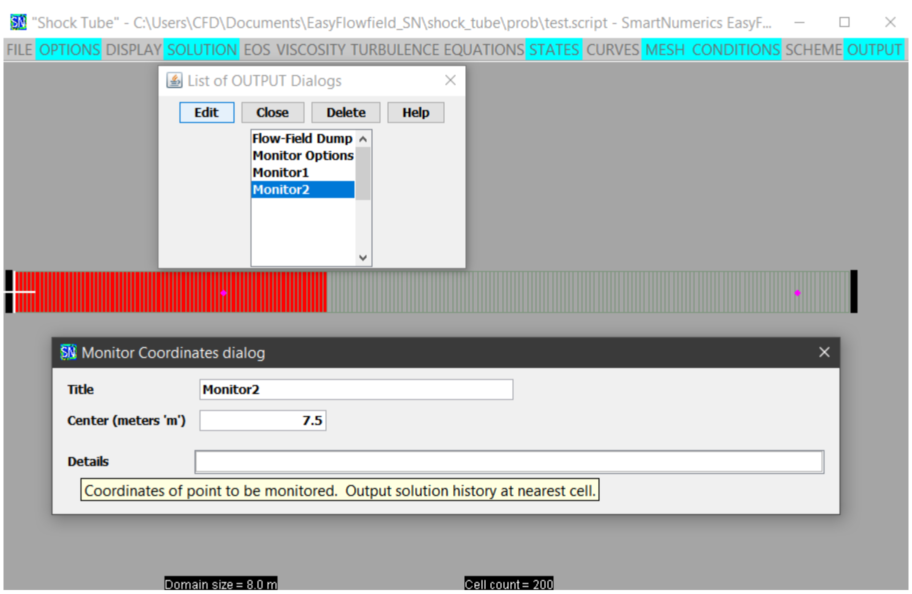


**Fig. 9: Specify locations of monitor points.**

Next open a Monitor Coordinates dialog and locate the monitor point at 2 meters. Then use New to create a second monitor point, change the title to "Monitor2", and specify a location of 7.5 meters. The values will be output in the files Monitor1.csv and Monitor2.csv. After closing the monitor dialogs, save the script using Save Script under menu heading **FILE**. Try holding down the Ctrl key on your keyboard and clicking on or near the left monitor point and on the left boundary. Labels will appear with information.

If you have difficulty seeing the first monitor point, please add an additional state dialog with a pressure of 11 or 12 atmospheres. Then open the Flow Region Display dialog and select 'Pressure Display' instead of 'Default Display'.

Please run the simulation using Interactive Simulation or Automated Simulation under menu heading **FILE**. In either case, the output files will be in shock_tube/output.

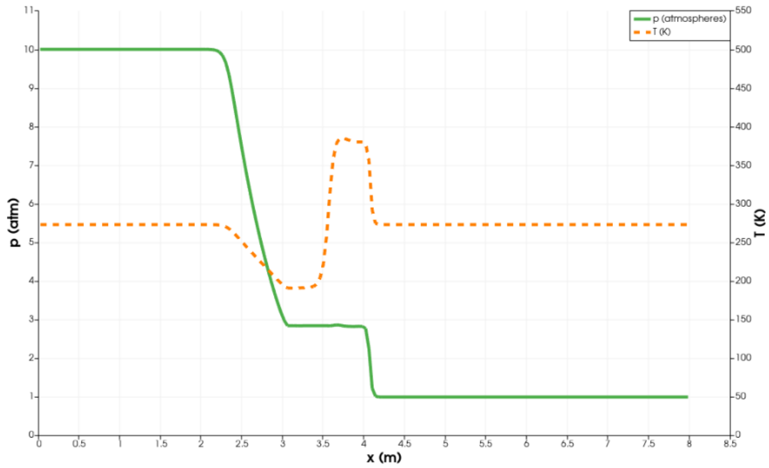


**Fig. 10: Pressure and temperature at 2 milliseconds.**

Figure 10, created using ParaView, displays the temperature and pressure at 2 milliseconds. A rarefaction wave is moving into the high-pressure gas and cooling it. A shockwave is moving into the ambient gas and heating it. There is a contact surface between the heated and cooled gas that moves at the post shock velocity. This is a fairly course grid. The shock and contact surface could be much thinner if more cells were used. This is practical for a one-dimensional simulation but less practical for more complex two- and three-dimensional simulations.

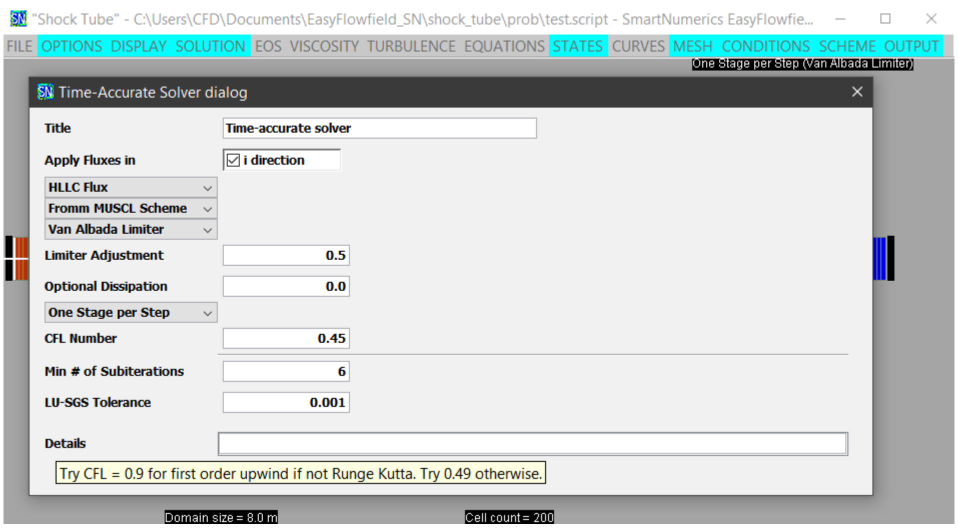


**Fig. 11: Time-Accurate Solver dialog.**

A thinner contact surface and shock front can be obtained as follows. Open the Time-Accurate Solver dialog under menu heading **SCHEME** and increase 'Limiter Adjustment' from 0.4 to 0.5. This is the maximum allowed value for most limiters. Please close the dialog, save the script, and rerun the simulation.

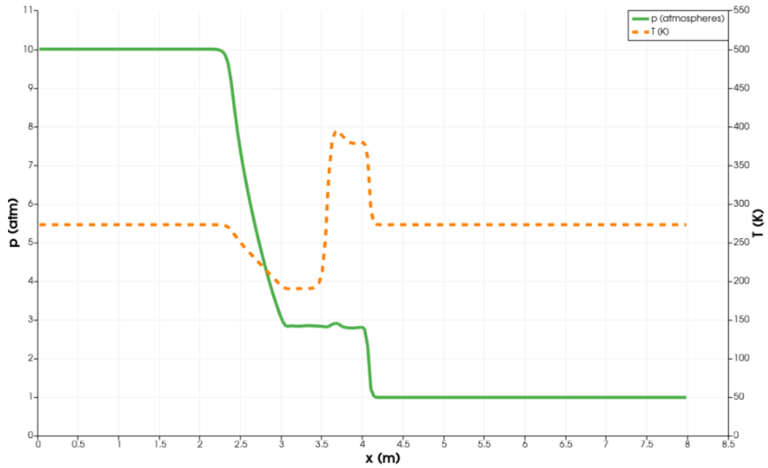


**Fig. 12: Pressure and temperature at 2 milliseconds with adjusted limiter.**

Figure 12 displays a sharper shock front and contact surface. However, the price of using a larger value of limiter adjustment is the presence of more wiggles behind the shock front. The wiggles could be reduced by setting 'Optional Dissipation' to 2 in the Time-Accurate Solver dialog.

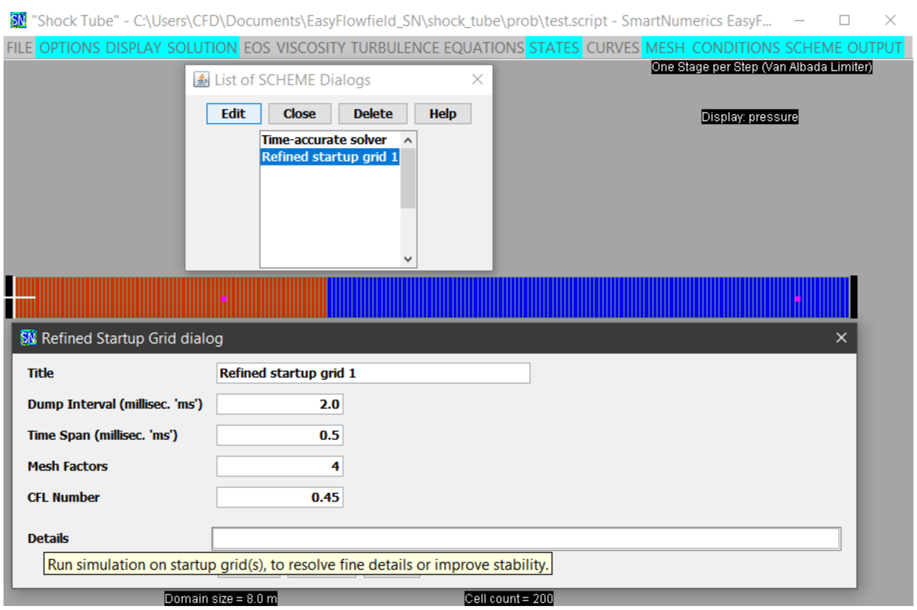


**Fig. 13: Using initial fine grid.**

Notice that the largest pressure bump is located near the contact surface. This can be reduced in size by using a finer grid at early times. To do this, open the Refined Startup Grid dialog under menu heading **SCHEME** and specify a time span of 0.5 milliseconds. Also specify a dump interval of 2 milliseconds to prevent output at 0.5 milliseconds. Set the Mesh Factor to 4 so that the startup grid will have 800 cells. Please close the dialog, save the script, and rerun the simulation.

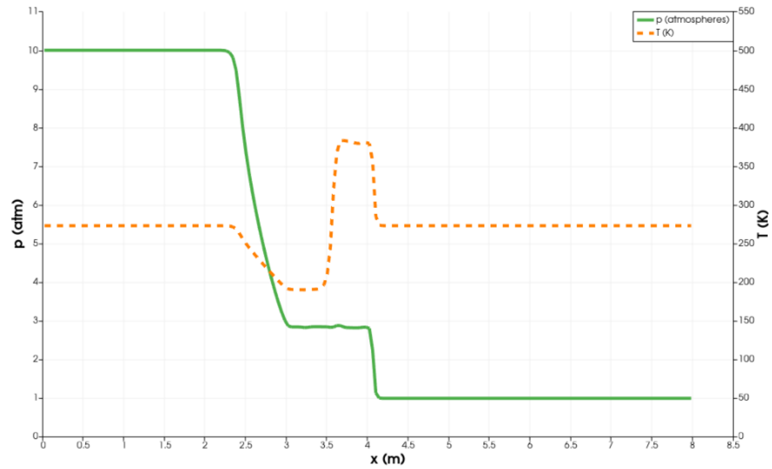


**Fig. 14: Pressure and temperature at 2 milliseconds with adjusted limiter and initial fine grid.**

Figure 14 displays the resulting pressure and temperature profiles. Not only has the pressure bump near the contact surface has been much reduced and an overshoot in temperature has been almost eliminated. 'Optional Dissipation' was not applied.

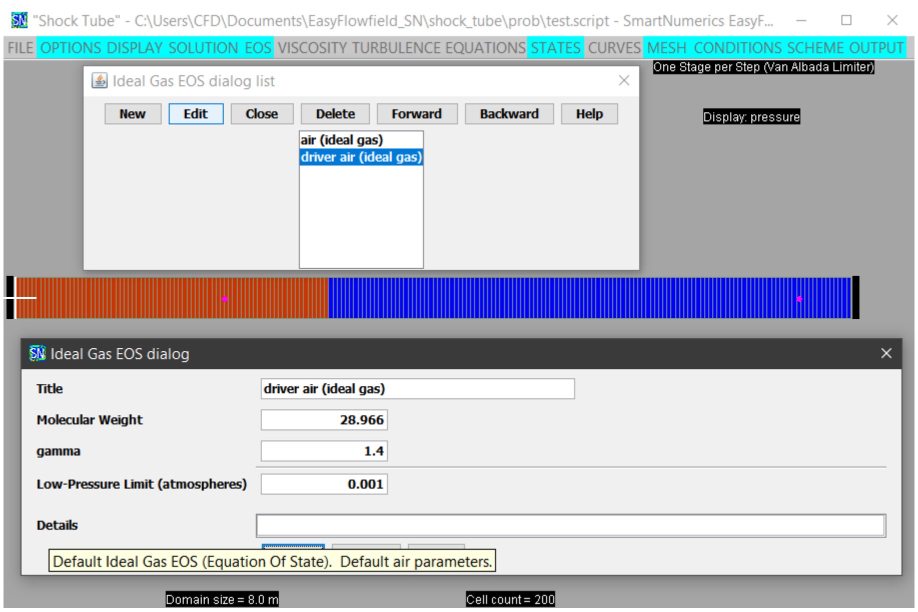


**Fig. 15: Dialog used to define ideal gas equation of state.**

By default, the solver uses the ideal gas equation of State (EOS) for air. Up to three equations of state can be used by the solver. In this case, two equations of state are used to trace the motion of the contact surface. First open and close an Ideal Gas EOS dialog under menu heading **EOS** without making any changes. Second use New to create a second Ideal Gas EOS dialog and change the title to "driver air (ideal gas)" and close it with no other changes. The second dialog should appear as displayed above. Note that the ideal gas EOS for a gas other than air can obtained by changing the molecular weight and the specific heat ratio (gamma).

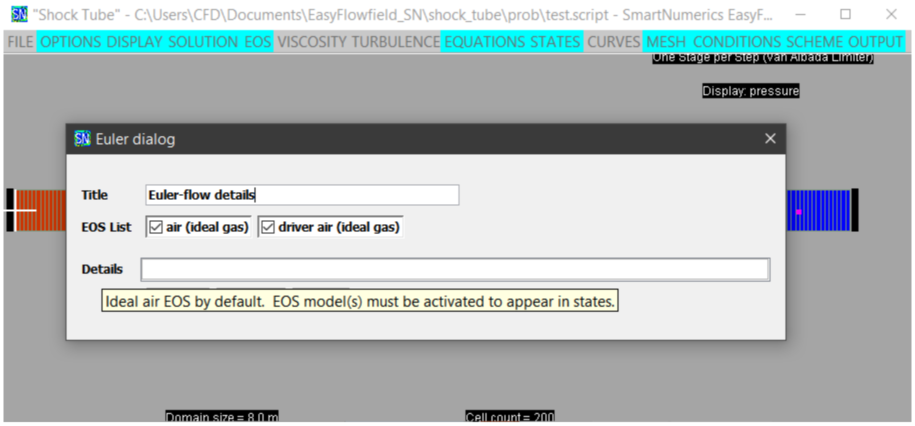


**Fig. 16: Euler dialog with both EOS checkboxes activated.**

Next open and close the Euler dialog under menu heading **EQUATIONS**. Both EOS checkboxes should be active for this simulation. Then reopen the driver-pressure Velocity State dialog under menu heading **STATES** and set the initial EOS to driver air (ideal gas). Please close the dialog, save the script, and rerun the simulation.

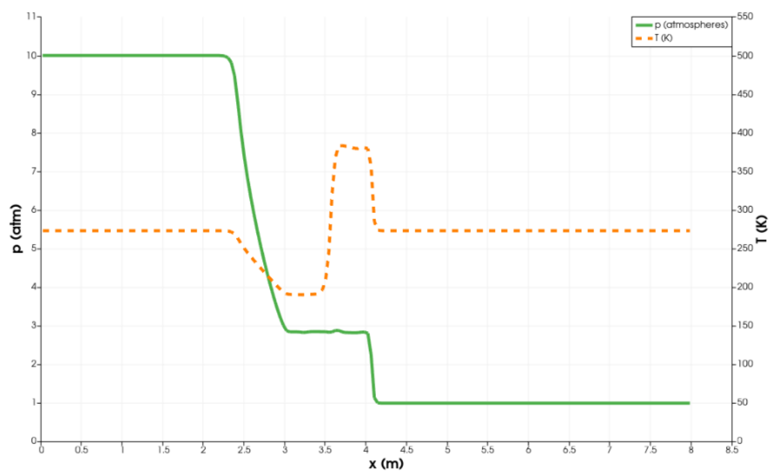


**Fig. 17a: Pressure and temperature at 2 ms with adjusted limiter, initial fine grid, and EOS tracer.**

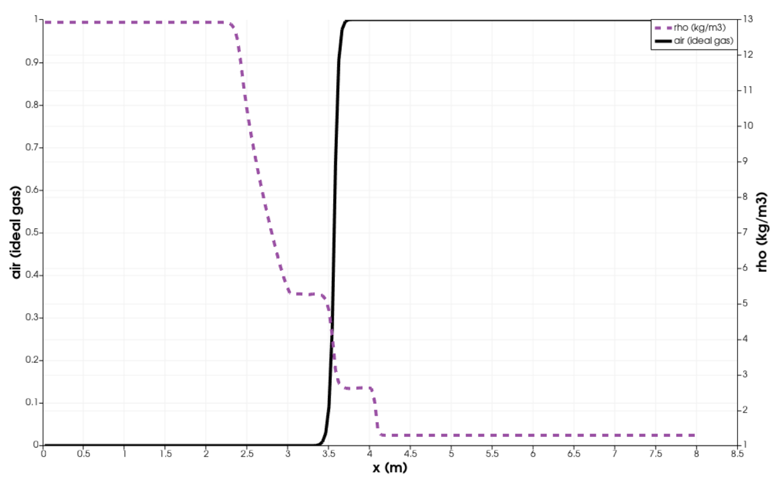


**Fig. 17b: Ambient air EOS fraction and density a 2 ms with adjusted limiter and initial fine grid.**

Figure 17a displays profiles of pressure and temperature at 2 milliseconds. Since both, the driver and ambient equations of state use the same parameters, the curves are identical to those obtained with only one EOS. Figure 17b displays profiles of density and the ambient air EOS fraction. As expected, there is a jump in density near the contact surface because the gas was cooled on the left and heated on the right of the contact surface.

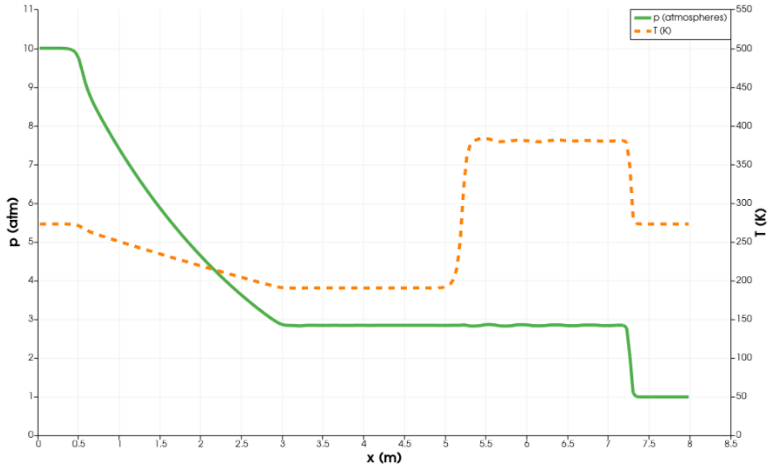


**Fig. 18a: Pressure and temperature at 8 ms with adjusted limiter, initial fine grid, and EOS tracer.**

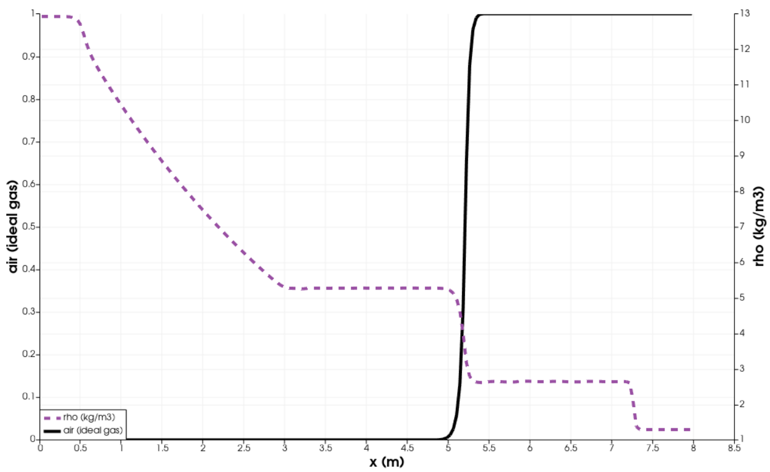


**Fig. 18b: Ambient air EOS fraction and density at 8 ms with adjusted limiter and initial fine grid.**

Figures 18a and 18b display the solution at 8 milliseconds. The contact surface will increase in thickness with time due to numerical diffusion, but the thickness of the shockwave does not increase.

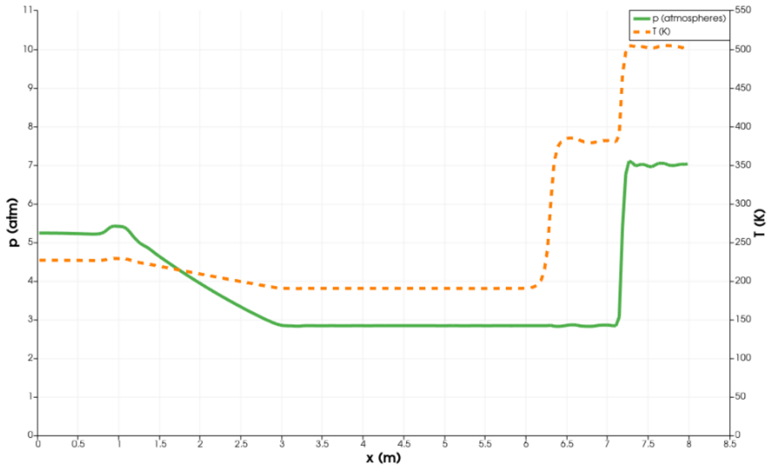


**Fig. 19a: Pressure and temperature at 12 ms with adjusted limiter, initial fine grid, and EOS tracer.**

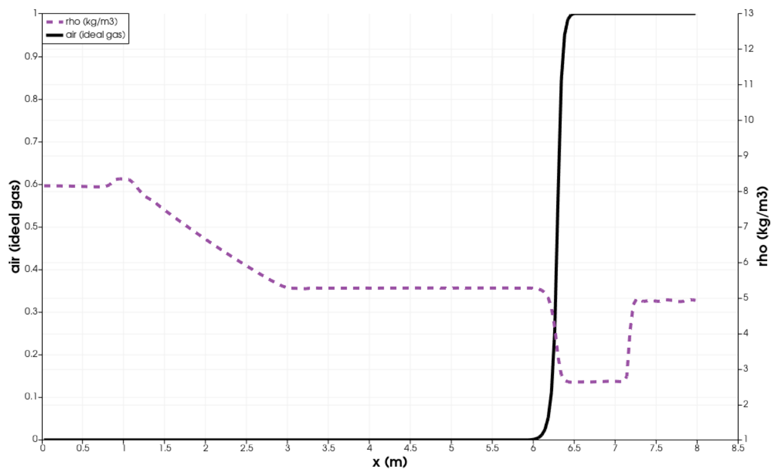


**Fig. 19b: Ambient air EOS fraction and density a`2 ms with adjusted limiter and initial fine grid.**

Figures 19a and 19b display the solution at 12 milliseconds. The shockwave has reflected of the right wall and is heading back towards the contact surface. The rarefaction wave is in the process of reflecting off the left wall.

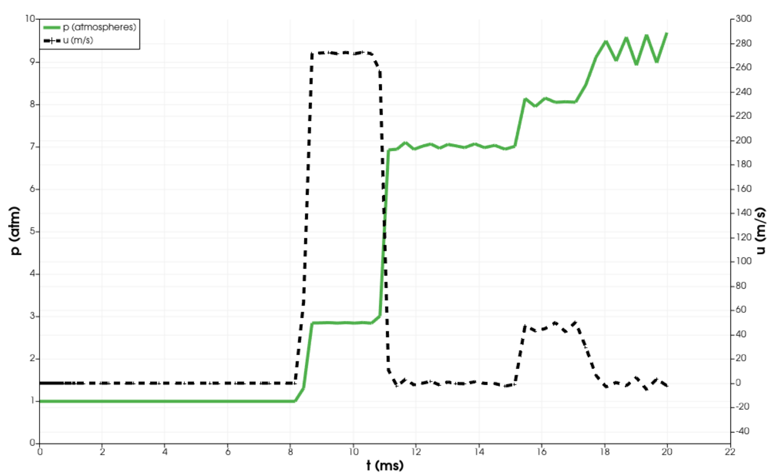


**Fig. 20: History of pressure and velocity at monitor 2 with adjusted limiter and initial fine grid.**

Figure 20 displays the history of pressure and velocity in the cell nearest to the location of monitor 2. The shockwave arrives just after 8 milliseconds. The reflected shockwave arrives just after 11 milliseconds. The reflected shockwave eventually passes through the contact surface which produces one more incident shockwave. This shockwave also reflects off the end of the shock tube.

You should now try using a smaller value of limiter adjustment such as 0.44 using the Time-Accurate Solver dialog to see the effect on the contact surface and shockwave at various times. The monitor 2 history will also be markedly improved. Try using different limiters or the first order upwind flux. Try doubling the number of points. These exercises will also give you a feeling for the effects on a two- or three-dimensional simulation.

Try changing Planar Geometry to Spherical Geometry in the Time-Accurate Solution dialog under menu heading **SOLUTION**. This will result in the simulation of the expansion of a spherical high-pressure region into an ambient region enclosed by a spherical wall. The outward moving shockwave eventually reflects off the wall. You can remove the reflection at the wall by reopening the Stretched Cartesian dialog under menu heading **MESH** and setting the right-hand boundary condition to Pass. Try doubling ‘Number of Cells’, ‘Grid-Block Size’, and ‘End Time’. An inward facing shockwave eventually develops near the origin and reflects off the origin. You will obtain a warning about negative pressure near the origin during this implosion process. However, the solver handles this issue, which in this case is relatively minor and transient. Reducing ‘Low Pressure Limit’ in the EOS dialogs only prevents warnings about miniscule positive pressures. **However, if you select ‘Runge-Kutta 2 Stage’ instead of ‘One Stage per Step’ in the Time-Accurate Solver dialog, the negative pressure transient will not occur.** Next, try setting ‘Grid-Layer Size’ to a quarter of ‘Grid-Block Size’ so that the cells are concentrated on the left side of the grid. Changing ‘Spherical Geometry’ to ‘X is Radial’ will result in the simulation of the expansion of a cylindrical high-pressure region.