EasyFlowfield Tutorial 6: Diffraction of a Shockwave
SmartNumerics Simulation Solutions Inc.

Version 6
June 5, 2020

Is this example, you will simulate a shockwave incident on an obstacle. The shockwave will reflect from the obstacle as well as diffract around it. This tutorial covers the construction and use of irregular objects immersed in a grid. This approach may be used with inviscid flow when simulating the impingement of propagating shock waves in complex environments. This tutorial also demonstrates the application of a post shock initial condition. Please read the shock-tube tutorial first.

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 a Simulation Overview dialog and change the title to "Shock Diffraction".

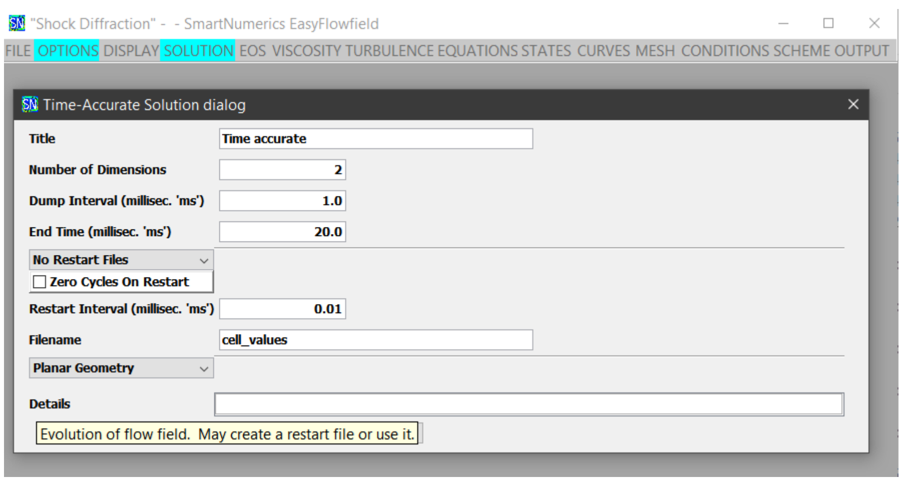

Fig. 1: Dialog used to control simulation of transient flow.

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

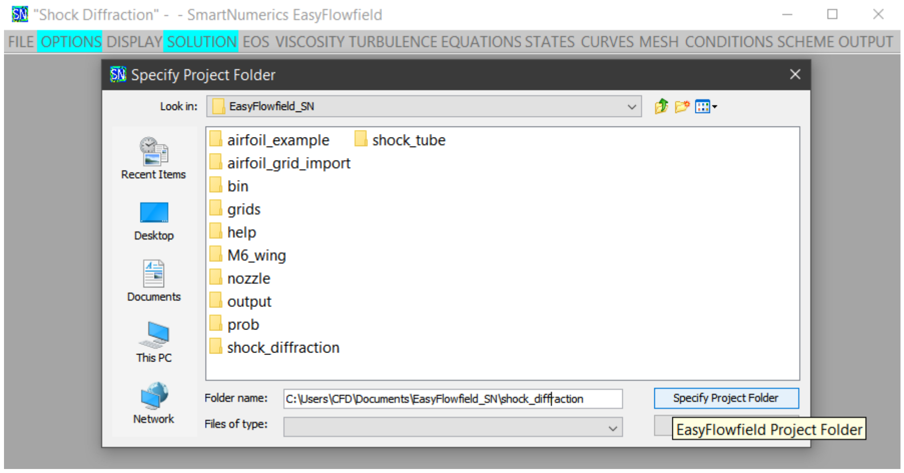

Fig. 2: Create project folder.

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

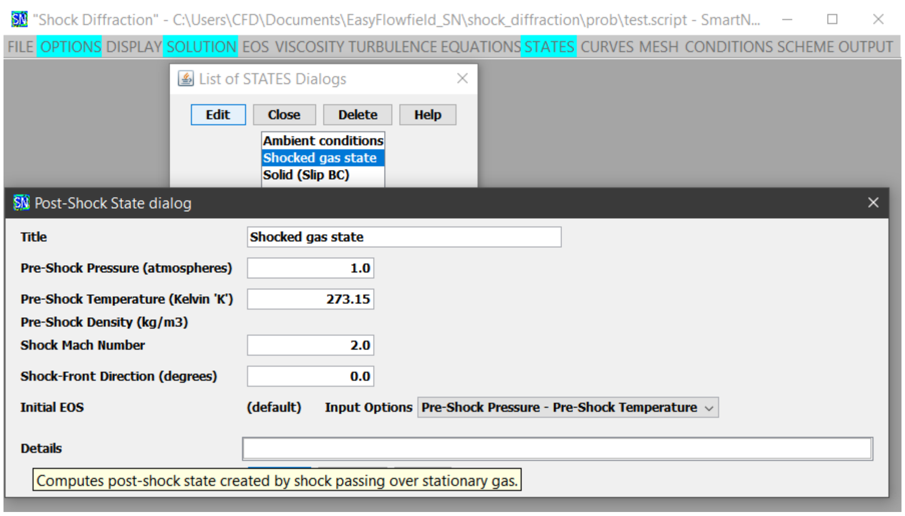

Fig 2: List of created state dialogs and post-shock state dialog.

To save the script, click on Save Script under menu heading **FILE**, navigate to shock_diffraction/prob, enter the filename "test", and click on 'Save Script File'.

Then open and close a Velocity State dialog under menu heading **STATES** without changing any parameters. Next open and close a Solid-State dialog without changing any parameters. Next open a Post-Shock State dialog and set the Shock Mach number to 2. This dialog will create post-shock conditions such that the shock will travel at twice the speed of sound of the gas ahead of the shock. The GUI with listed states should appear as shown above. Please save the dialog periodically to preserve your work.

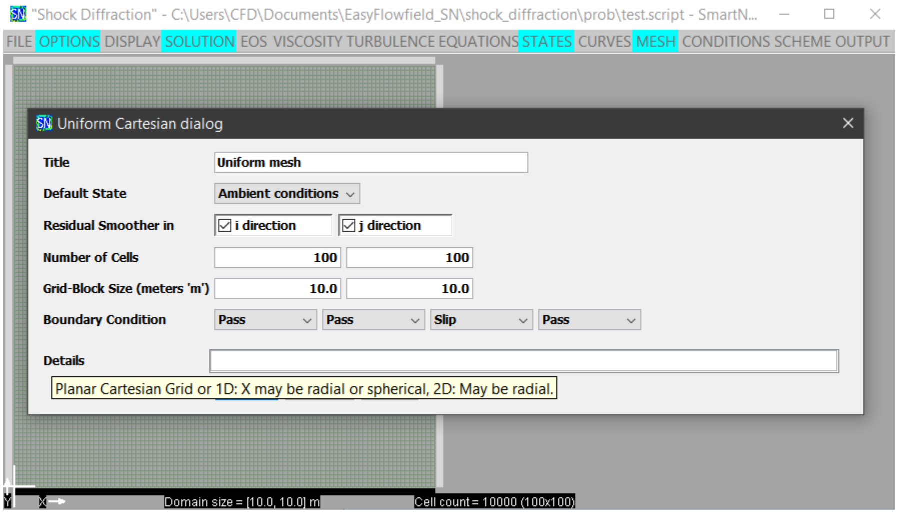

Fig. 3: Parameters used to generate grid.

Next open the uniform Cartesian dialog under menu heading **MESH** and specify a Slip boundary condition along the bottom of the grid. Also set the number of cells in each direction to 100 and the x and y dimensions of the grid to 10 meters. In the figure above, the Slip boundary is marked by a thick black line while all the other boundaries remain at the default Pass condition and are marked by thick white lines.

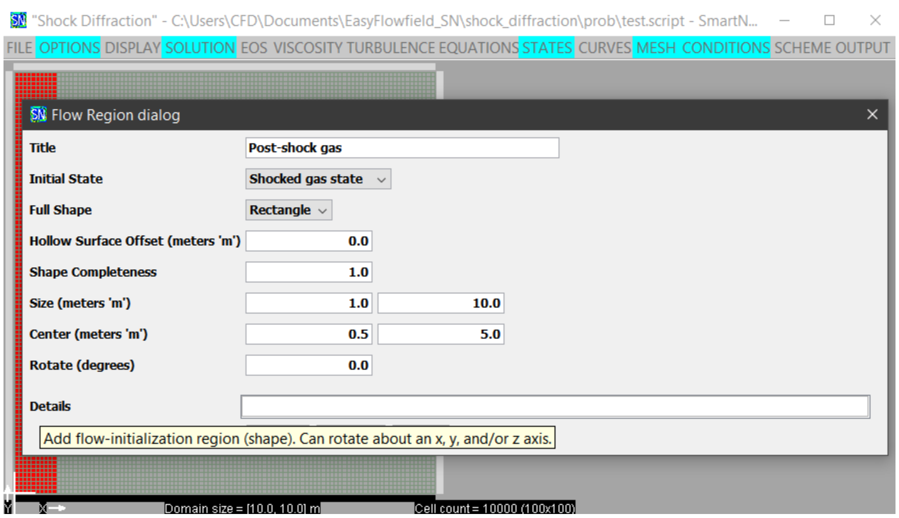

Fig. 4: Creation of initial post-front region.

Next open a Flow Region dialog, set the title to "Post-shock gas", set 'Initial State' to the shocked gas state and specify a rectangle of 1 by 10 meters centered at [0.5, 5] meters. Close the dialog.

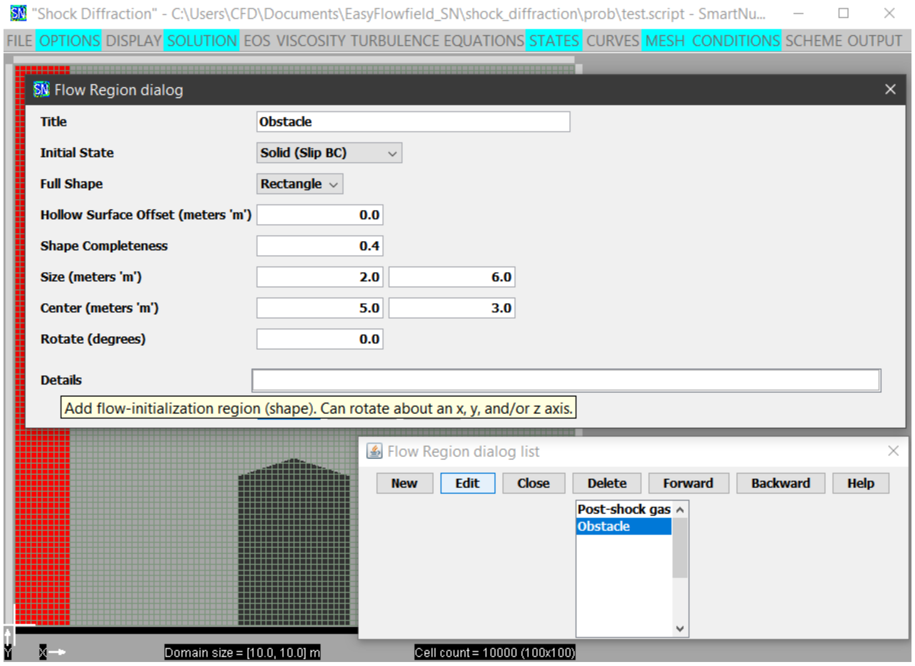

Fig. 5: Creating obstacle ahead of shockwave.

Next, use New to create a second Flow Region dialog, set the title to "Obstacle", and specify a rectangle of 2 by 6 meters centered on [5, 3] meters. Also set the initial state to Solid (Slip BC) and set Shape Completeness to 0.4 to the give the solid region a more complex shape. Extremely complex shapes for the Flow-Region can be input from a text file as detailed in help for the Flow-Region Outline dialog under menu heading **CURVES**.

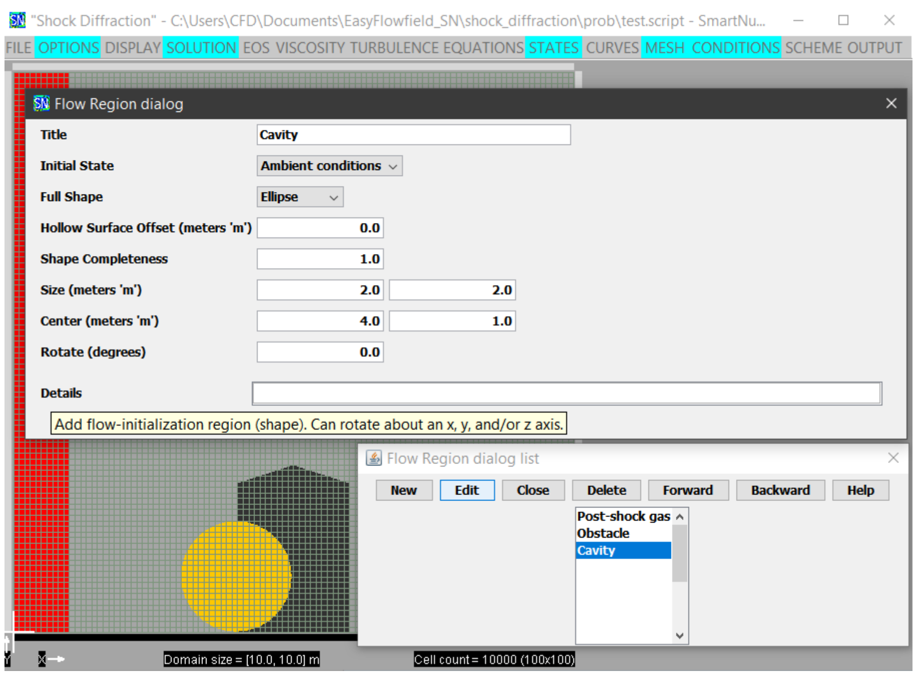

Fig. 6: Creating cavity at front face of the obstacle.

To make the obstacle even more complex, please create one more Flow Region dialog, set the title to "Cavity", and specify an ellipse (circle) with a diameter of 2 meters centered at [4, 1] meters. The ambient conditions should be used for the initial state. Please close the dialog. Try holding down the Ctrl key on your keyboard and clicking on the various flow regions and edges of the grid. A label will appear with information.

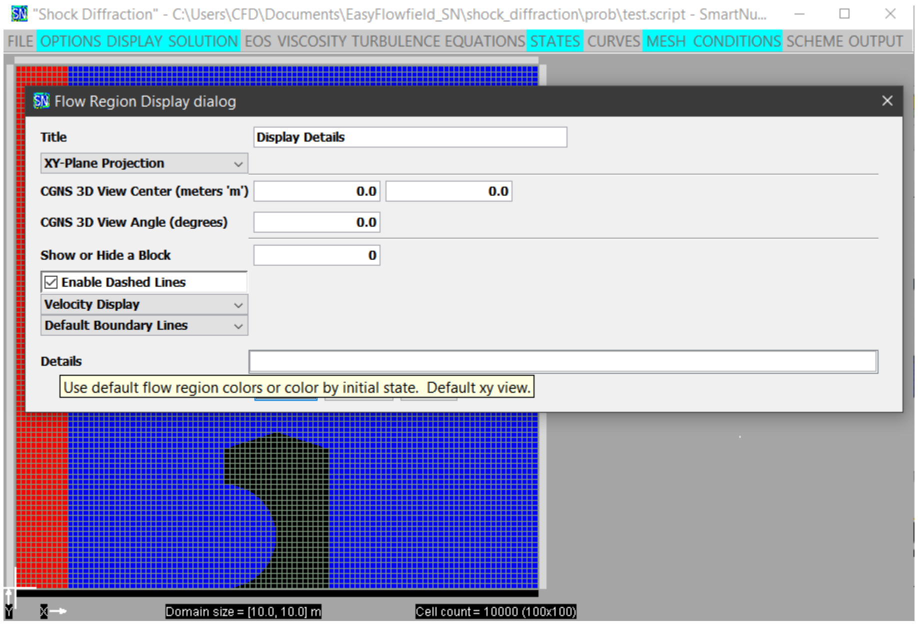

Fig. 7: Setting display of flow regions.

Next open the Flow-Region Display dialog under menu heading **DISPLAY**, change Default Display to Velocity Display, and close the dialog. The obstacle should appear in black as shown above. The high velocity post shock gas is red, and the zero-velocity ambient gas is blue. Try moving the Cavity region 'forward' and 'backward' in the Flow Region dialog list to see the effect on the obstacle. Try hitting the '- ' and '+' keys on the keyboard to make the grid disappear and appear. Try replacing Default Boundary Lines by Thin Boundary Lines in the Flow Region Display dialog and closing the dialog. Please close the dialog and save the script using Save Script under menu heading **FILE**. Next, use Save Script As to save the script as test_3D.script.

Please try changing the number of dimensions in the Time-Accurate Solution dialog to 3. The 3D flow regions will then be displayed using wireframe coloured by the velocity corresponding to the state. The shape of the obstacle is obscure in 3D unless you select 'Default Display' and 'Thin Boundary Lines' in the Flow-Region Display dialog. Open Rotate Mesh under menu heading **OPTIONS** and enter a rotation of [9, 9, 0] degrees to see a counter rotation of the wireframe outlines of the flow regions. Please save test_3D.scipt and re-open test.script using Read Script under menu heading **FILE**.

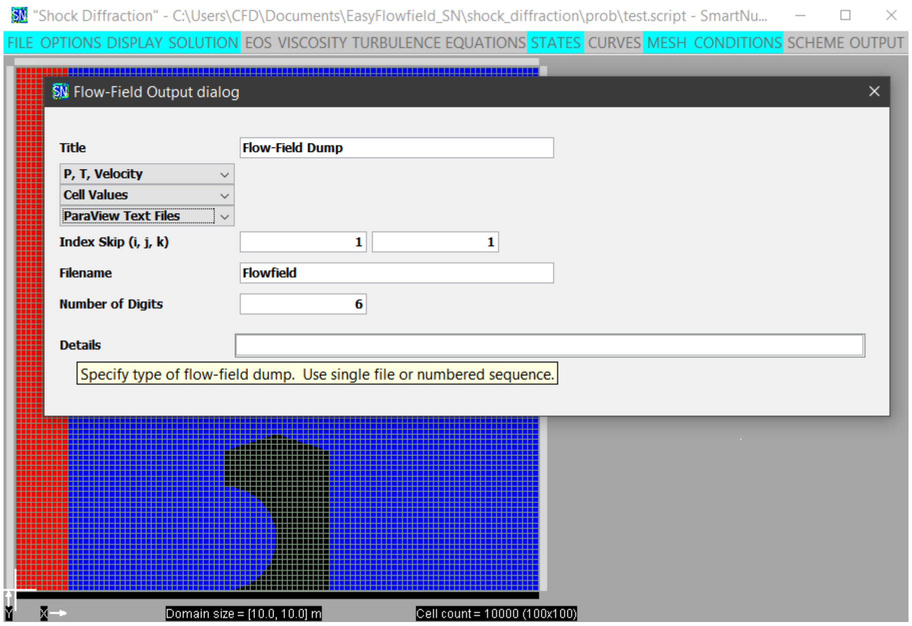

Fig. 8: Selection of flow-field output.

Next, open the Flow-Field Output dialog under menu heading **Output** and select 'P, T, Velocity' to output values of pressure, flow speed, and velocity components in each cell. Also select the 'ParaView Text Files' format (.vts).

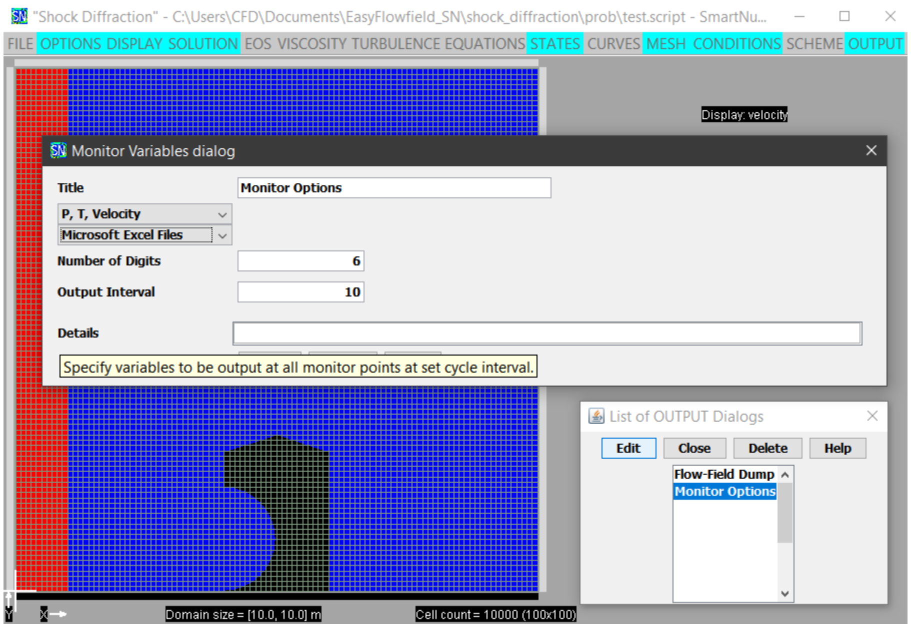

Fig. 9: Specify monitor variables and output file format.

Next open the Monitor Variables dialog and select 'P, T, Velocity' to output values of Pressure, temperature, and the components of flow velocity at specified locations. Also select the Microsoft Excel Files format. Values will be output in text files using the "comma separated values" format (.csv).

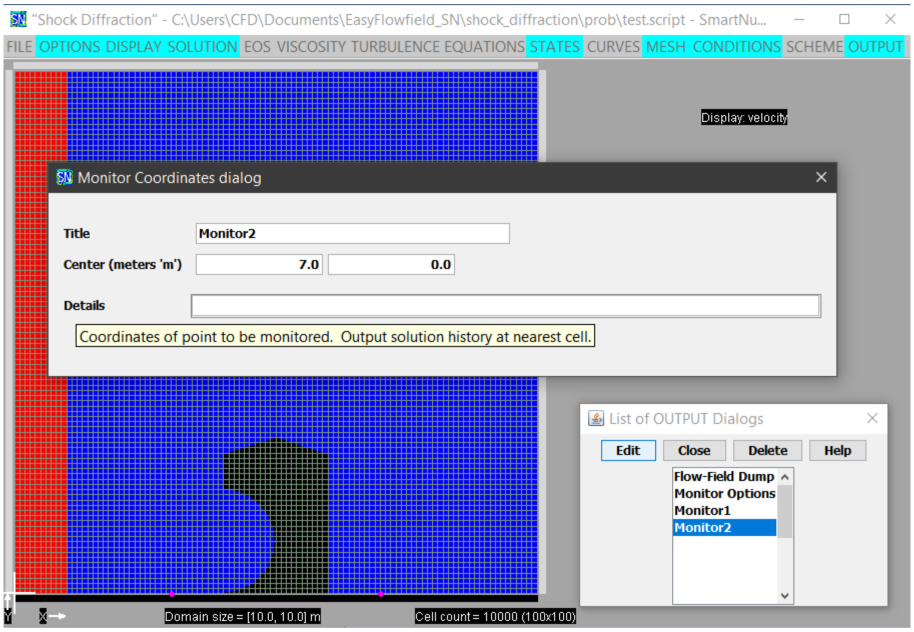

Fig. 10: Specify monitor locations.

Next open a Monitor Coordinates dialog and specify a center of [3, 0] meters. The values of pressure, temperature, and the velocity components in the cell nearest to [3, 0] meters will be saved in a text file with the name Monitor1.csv. Then use New to open a second Monitor Coordinates dialog, rename it to "Monitor2" and specify a center of [7, 0] meters. After closing the monitor dialogs, save the script using Save Script under menu heading **FILE**.

Finally open the Rescaling Factors dialog under menu heading Options and set ‘Mesh Multiplier’ to 3 to produce a 300x300 cell grid. You should disable display of the gridlines so the obstacle will be visible.

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

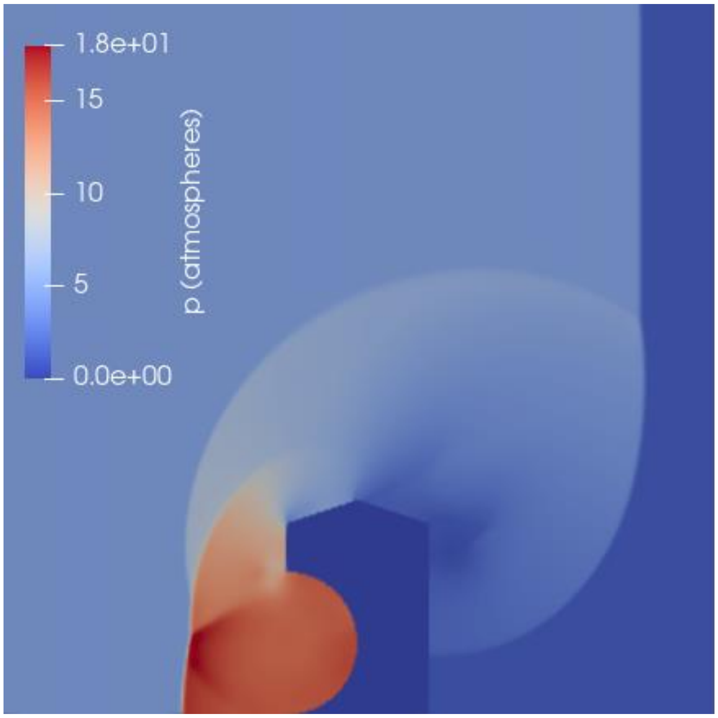

Fig. 11: Pressure contours at 12 milliseconds produced by ParaView.

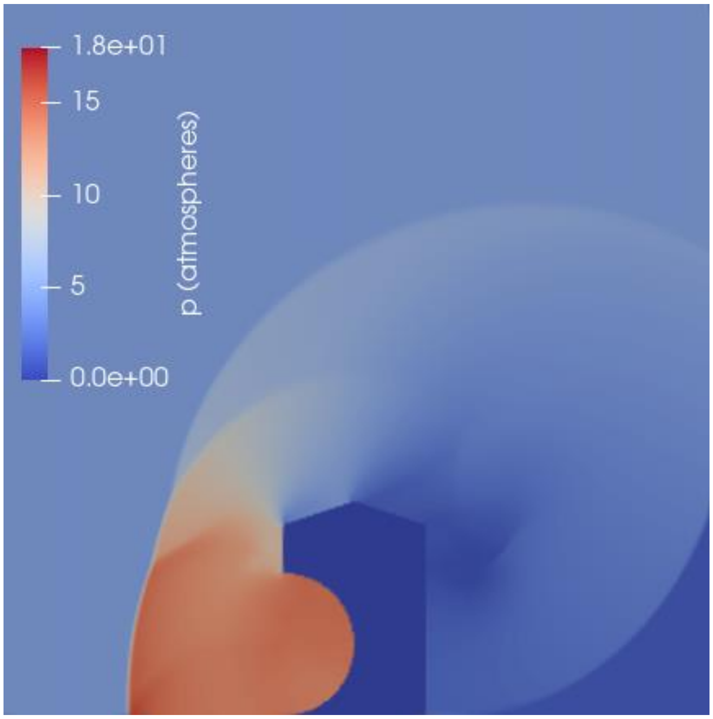

Fig. 12: Pressure contours at 14 milliseconds produced by ParaView.

Figures 11 and 12 display the pressure contours at 12 milliseconds, and 14 milliseconds, respectively. The pressure and temperature histories at the monitor points are displayed in Figures 13 and 14. The speed of sound in ahead of the shock is 331 m/s. A Mach 2 shock travels at twice the speed of sound. The distance from the initial shock front to the first monitor point is 2 meters. This distance, divided by the speed of the shock front, is about 3 milliseconds. This is when the center of the shock front is seen to arrive in Figure 13.

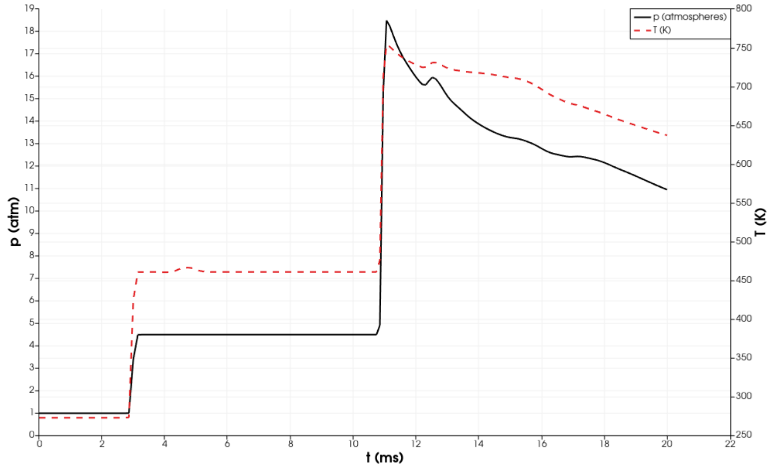

Fig. 13: Pressure and temperature history for monitor point 1.

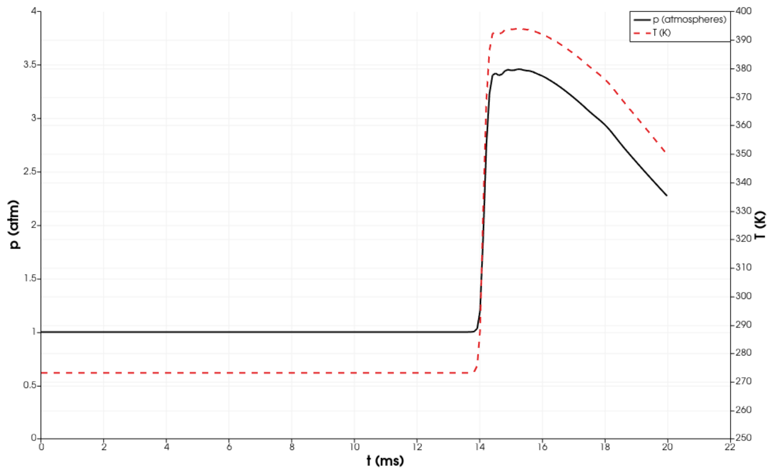

Fig. 14: Pressure and temperature history for monitor point 2.

Please try opening the Time-Accurate Solver dialog under menu heading **SCHEME** and setting ‘Limiter Adjustment’ to 0.5. Also select ‘Runge-Kutta 2 Stage’ instead of ‘One Stage per Step’. See how this effects the pressure contours and monitor history. Try doubling the number of cells in the streamwise direction.