## Virtual Wind Tunnel, Axisymmetric Convergent Divergent Nozzle

### Brief description

This simulation acts as a “virtual wind tunnel” by allowing students to simulate a converging diverging nozzle under different chamber and ambient pressures while also allowing students to change the overall shape of the nozzle.

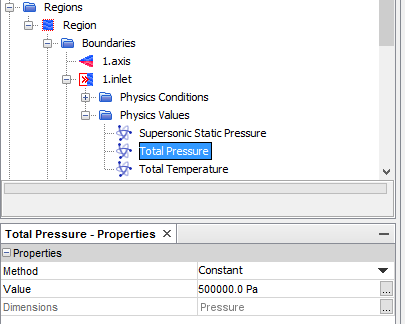
### Learning objectives

At the end of this exercise you should be able to ….

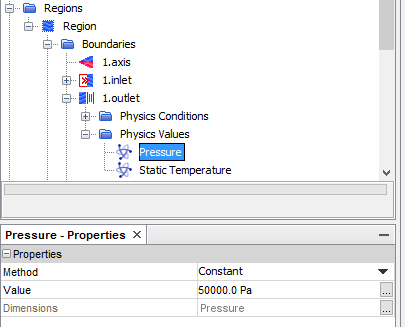
1. Understand the cause/affect relationships between shock formations, pressure ratio per along the walls, and the overall solution fields (Mach, temperature, density, and velocity) resulting from changes in:
   1. Chamber and ambient pressure
   2. Nozzle geometry (throat radius, exit radius, etc)
2. Understand the difference between inviscid and viscous/turbulent solutions.

### Steps you need to take

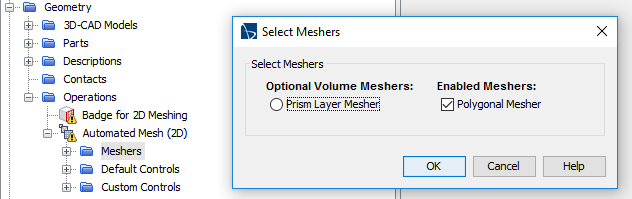
1. Open STAR-CCM+.
2. File, load simulation.
3. Where appropriate, enter additional licensing options if needed/wanted (Power on Demand, put in your PoD key, select parallel, etc.).
4. Browse, go to the folder containing the sim file for this course, select virtual\_wind\_tunnel\_axisymmetric\_compressible\_CD\_nozzle. Select Open, OK. Please note that this simulation is setup as inviscid. We will be covering how to covert this to a viscous/turbulent simulation in step 14.
5. Select the Create/Open Scenes icon at the top of the screen (), then select Open All Scenes. Note that multiple tabs will open above, you can click these tabs to switch between what is shown in the viewing window.
6. To manipulate the viewing of a scene:
   1. Panning: To pan the image in the visualization display, hold down the right mouse button and drag the mouse in the direction where you want to move the object. The image appears to change its view angle as you pan. This effect is because the view is in the Perspective projection mode, which depicts three-dimensional space. In the Parallel projection mode, the view angle stays the same.
   2. Rolling: To roll the scene, hold down the <Ctrl> key, and click and drag the mouse upward or downward. Using the <Ctrl> key rotates the object along an axis perpendicular to your screen.
   3. Rotating (not enabled for 2d simulations): STAR-CCM+ uses a point on a part to rotate. If you select a point somewhere other than on a part, STAR-CCM+ uses the depth to that last point; the depth is normal to the view plane. The scene can be rotated around various axes, vertical, horizontal, or arbitrary, depending on the direction you move the mouse. To rotate, choose a point on or near the object in the visualization display, and click and drag (holding down the left mouse button) from that point. This technique rotates the object around the axis perpendicular to the direction of your drag. You set the location of the axis when you choose a point to begin clicking and dragging.
   4. Zooming: There are two ways to zoom with the mouse:
      * 1. Choosing a focal point: Choose a point inside the display, then click and drag downward using the middle mouse button. This action brings your view closer to that point. Moving the mouse straight up has the opposite effect.
        2. Using the mouse wheel: Turning the mouse wheel toward you brings your view closer to the visualization display; the opposite wheel movement takes your view farther away.
      1. The mouse pointer must be within the visualization display for this operation to work. Its particular position within the display determines the direction of this type of zooming.
   5. Hot Keys:
      1. Side View, S key
      2. Top View, T key
      3. Front View, F key
      4. Flip vertical, C key
      5. Reset View, R key
   6. Animations: Click  to play and  to stop. In this simulation, animations are enabled for the vector scenes only.
7. Select the Create/Open Plots icon at the top of the screen (), then select Open All Plots. These plots will help you to compare drag, lift, Cd, and Cl numerical results for the body in use. When changes are made to the body or freestream velocity, you will also be able to use these plots to numerically compare the results of the new analysis compared to the old one. For the purpose of this exercise you can ignore the residuals tab. To zoom in/out on the plots, use the middle mouse wheel or this icon . To reset the view hit R on the keyboard or the icon shown here in the center .
8. With the plots tabs now open at the top of the screen, there are a number of tabs to sort through. To see the tabs that are hidden to the left/right of the screen, use the arrows () at the top right of your screen.
9. If you would like to change the geometry shape, please read steps 10 and 11. If you would like to change the chamber or ambient pressure please read step 12.
10. To change the geometry shape, in the simulation tree expand Geometry, 3D-CAD Models, right click 3D-CAD Model 1, edit, right click sketch 2, edit. Then move the spline points for the outer nozzle as needed. To adjust the vectors for each spline, first click the spline. Then move the arrows as needed.
11. Once you have made the desired changes to the geometry, click the  icon at the top of the screen. Browse to the java file called adaptive\_mesh\_and\_run.java and select Open. This java macro will automatically clear the existing solution and will then run using an adaptive mesh. Now skip to step 13.
12. To change the chamber pressure, please go to Regions, Region, Boundaries, 1.inlet, Physics Values, and select Total Pressure. In the properties window below change the value as desired.



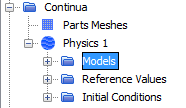
To change the ambient pressure, please go to Regions, Region, Boundaries, 1.outlet, Physics Values, and select Pressure. In the properties window below change the value as desired.



Once you have made the desired changes, click the  icon at the top of the screen. Browse to the java file called adaptive\_mesh\_and\_run.java and select Open. This java macro will automatically clear the existing solution and will then run using an adaptive mesh.

1. While the simulation runs, you will see the current results in the scenes and plots. The simulation should stop at 1100 solve iterations.
2. To change the simulation to viscous/turbulent, first go to Geometry, Operation, Automated Mesh (2D), and right click Meshers. Select “Select Meshers…”. Select Prism Layer Mesher, then select OK. 

Next go to Continua, Physics 1, right click Models, select “Select Models…”.



Uncheck Inviscid on the right. On the left, select Turbulent and K-Epsilon Turbulence. Click Close at the bottom of the screen. See steps 10, 11, and 12 if any changes need to be made to chamber or ambient pressure, or the geometry. If not, click the  icon at the top of the screen. Browse to the java file called adaptive\_mesh\_and\_run.java and select Open.

Disclaimer:

The focus of this lesson is on the approximate flow features. This simulation is meshed in 2d to allow a larger number of computers the ability to run these calculations.