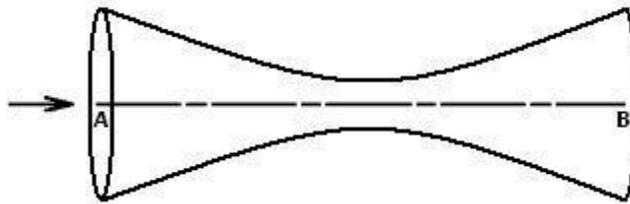


SPC 407
Supersonic & Hypersonic Fluid Dynamics
Ansys Fluent Tutorial 1

Compressible Flow in a Nozzle

Ahmed M Nagib Elmekawy, PhD, P.E.

Problem Specification



Consider air flowing at high-speed through a convergent-divergent nozzle having a circular cross-sectional area, A , that varies with axial distance from the throat, x , according to the formula

$$A = 0.1 + x^2; -0.5 < x < 0.5$$

where A is in square meters and x is in meters. The stagnation pressure p_0 at the inlet is 101,325 Pa. The stagnation temperature T_0 at the inlet is 300 K. The static pressure p at the exit is 3,738.9 Pa. We will calculate the Mach number, pressure and temperature distribution in the nozzle using FLUENT and compare the solution to quasi-1D nozzle flow results. The Reynolds number for this high-speed flow is large. So we expect viscous effects to be confined to a small region close to the wall. So it is reasonable to model the flow as inviscid.

Pre-Analysis & Start-Up

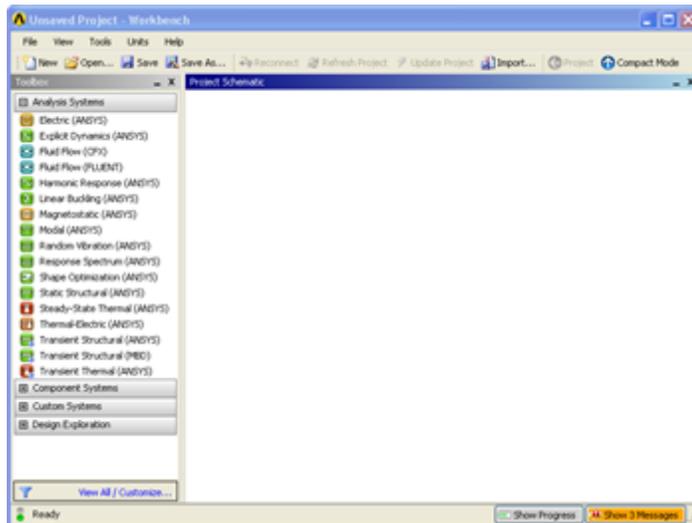
Hand Calculations

In quasi - one dimensional flow for this converging diverging nozzle, we expect that the Mach number at the inlet will be subsonic and accelerating until at the minimum nozzle area (the throat) the flow becomes sonic with $M = 1$. At that point, we expect that since the nozzle starts to diverge the flow will continue accelerating until at the exit, the flow continues as supersonic. We get to this result via the area velocity relation:

$$\frac{dA}{A} = (M^2 - 1) \frac{dV}{V}$$

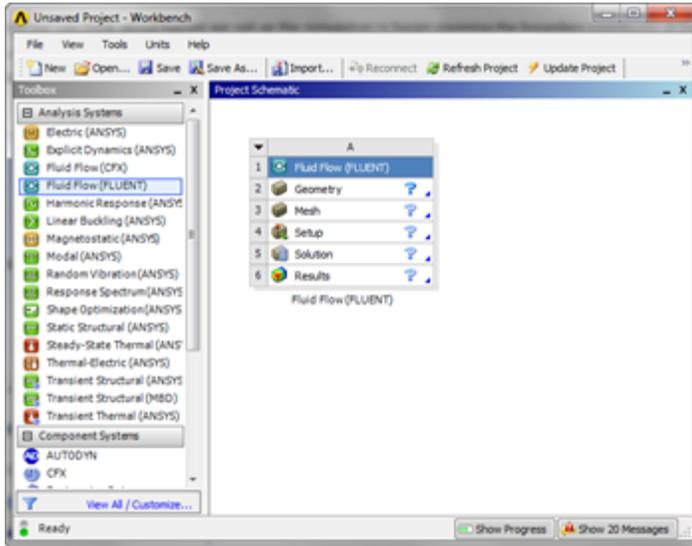
Open ANSYS Workbench

We are ready to do a simulation in ANSYS Workbench! Open ANSYS Workbench by going to Start > ANSYS > Workbench. This will open the start up screen seen as seen below



Setup Project

To begin, we need to tell ANSYS what kind of simulation we are doing. If you look to the left of the start up window, you will see the Toolbox Window. Take a look through the different selections. We will be using FLUENT to complete the simulation. Load the **Fluid Flow (FLUENT)** box by dragging and dropping it into the Project Schematic.



Right click the top box of the project schematic **1** **Fluid Flow (FLUENT)** and go to **Rename**, and name the project “Compressible Flow in a Nozzle”. You are ready to create the geometry for the simulation.

Step 1: Geometry

We only need to model half of the nozzle cross-section because it can be solved using the axisymmetric solver in FLUENT. The 2D axisymmetric geometry has 4 edges; we will need to import a coordinate file to create the diverging-converging nozzle wall. The coordinate file can be found in the tutorial archive.

Coordinates File

You should see five columns of data in the coordinates file. The first column is the **Group number**. The coordinates data are assigned to groups to distinguish edges.

The second column is the **Group ID**. Each set of coordinate needs to be assigned an ID so they can be imported in the correct order. The third column is the **x coordinate**. It varies from **-0.5** to **0.5**, as shown in the problem statement.

The fourth column is the **y coordinate**. The y coordinate (**radial coordinate**) can be written as the radius of the nozzle in terms of nozzle cross section area:

$$A = \pi * r^2.$$

for the given nozzle geometry, we get

$$r(x) = [(0.1 + x^2)/\pi]^{0.5}; -0.5 < x < 0.5$$

The last column is the **z coordinate**. Although we are working with a 2D model, the tool we will use to import the coordinates requires coordinates in all three directions. We can conveniently set the z coordinates to **0**. The coordinates in the text file need to be grouped in a way that the end coordinate of a group must be the start coordinate of the next group. For instance, the end coordinate of group 1 is (0.5 0.333778669, 0) is the same as the start coordinate of group 2.

Create the Geometry

Watch the tutorial video.

A summary of the video steps is shown below:

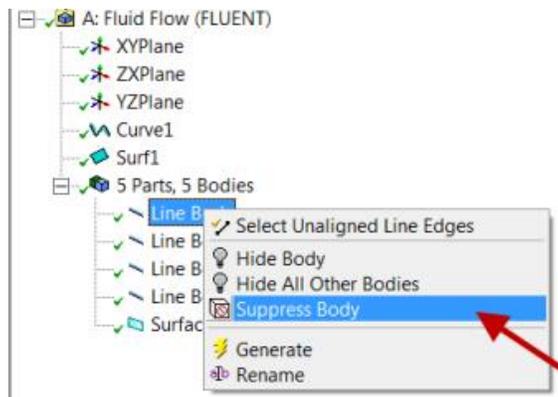
1. Drag Fluid Flow (FLUENT) to the Project Schematic
 - a. Rename it 2D Nozzle
2. Right click on Geometry > Properties
 - a. Analysis Type > 2D
3. Right click on Geometry > New Design Modeler Geometry, to open up Design Modeler
4. Select Meters as Units
5. Go to Top Menu, click on Concept > 3D curve
 - a. For Coordinate File > Browse and select nozzle coordinates.txt
6. Click on the Zoom to Fit icon
7. Click on Concept > Surfaces (Edges)
 - a. Click on all four edges
8. Generate

Note that while following the video above, you can select the 4 edges to create a surface in two ways:

1. By holding down the *Ctrl* key and clicking on the 4 edges in turn.
2. By "painting" over the 4 edges i.e. dragging the left mouse button over the four edges. This feature is called *paint select*.

Suppress line bodies

You need to suppress the line bodies before moving on, otherwise you might get an error later. Right click on each line body and select **Suppress Body** as shown below. Then, save your project before moving to the next step.



Since the nozzle has a circular cross-section, it's reasonable to assume that the flow is axisymmetric. So the geometry to be created is two-dimensional.

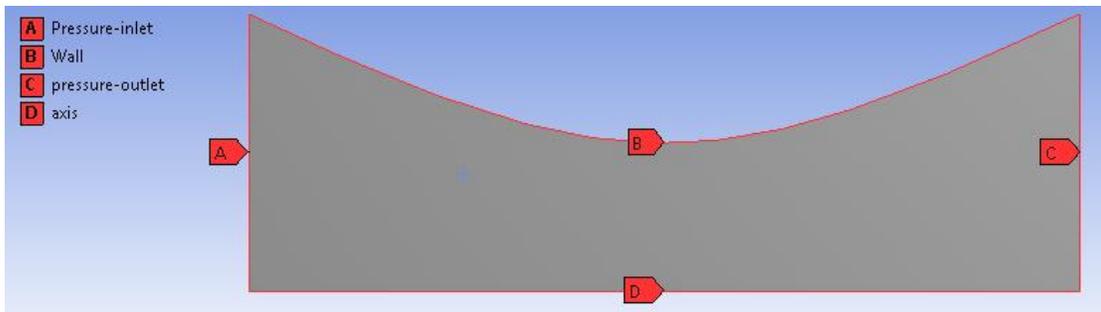
Step 3 Mesh

Watch the tutorial video.

A summary of the video steps is shown below:

1. Right click on Mesh > Insert > Face Meshing
2. Click on face of the geometry > Apply
3. Right click on Mesh > Insert > Sizing
4. Click on Select Body icon > click on geometry > Apply
5. Under Element Size enter 0.025 m
6. Generate

After creating the surface, name each of the edges of the surface. Do this by using the edge selection tool, right clicking on the line that you want to name, selecting 'Create a Named Selections' and typing in your desired name, then click the 'Generate' symbol to generate that named selection. You will want to name the pressure inlet, the outlet, the nozzle wall, and the axis.

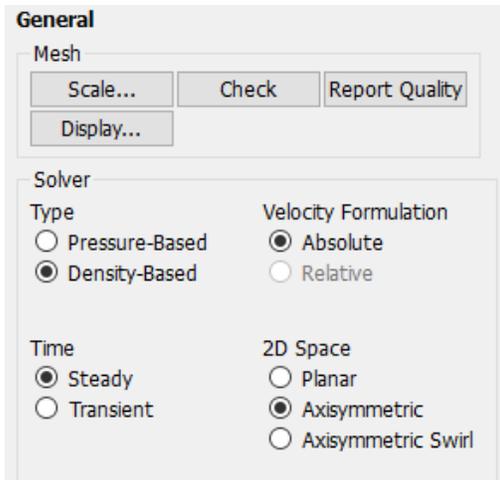


Step 4: Set Up Problem in FLUENT

Define Properties

General

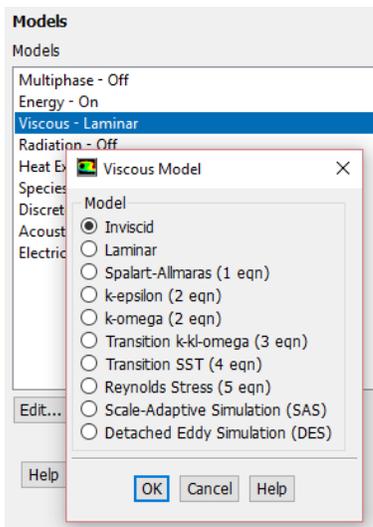
Under the General box, select *Density-Based*. Under 2D Space, choose *Axisymmetric*.



Click OK.

Models > Viscous

Select *Inviscid* under Model.

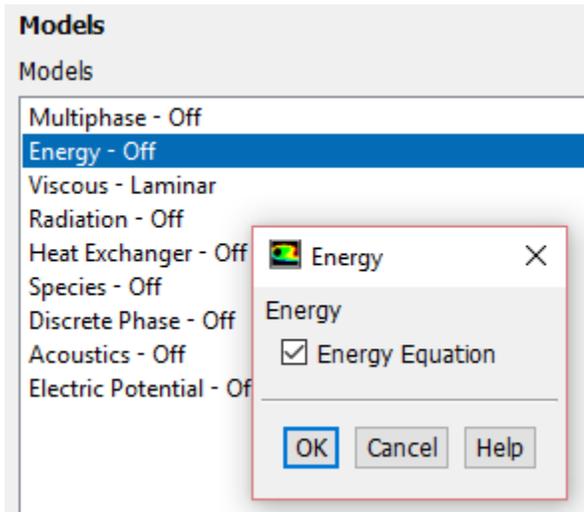


Click OK.

Define > Models > Energy

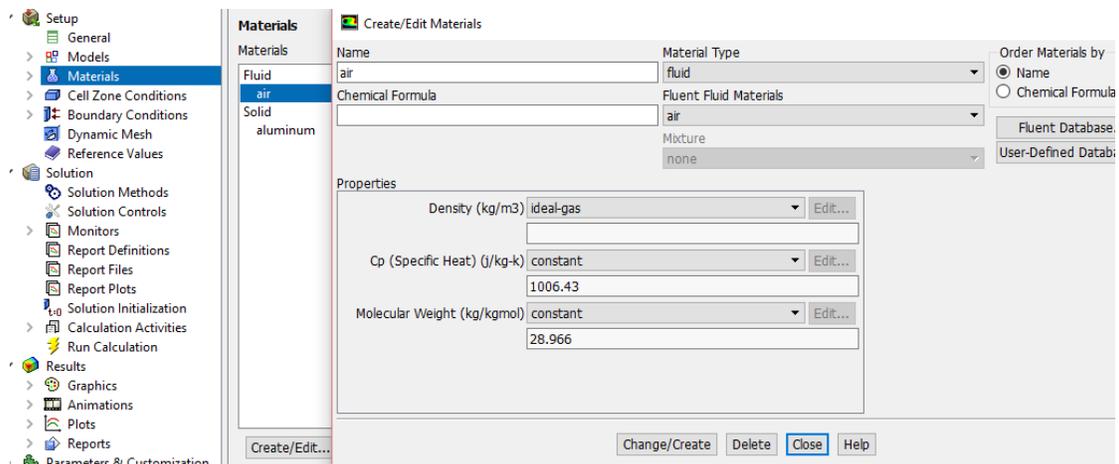
The energy equation needs to be turned on since this is a compressible flow where the energy equation is coupled to the continuity and momentum equations.

Make sure there is a check box next to Energy Equation and click OK.



Define > Materials

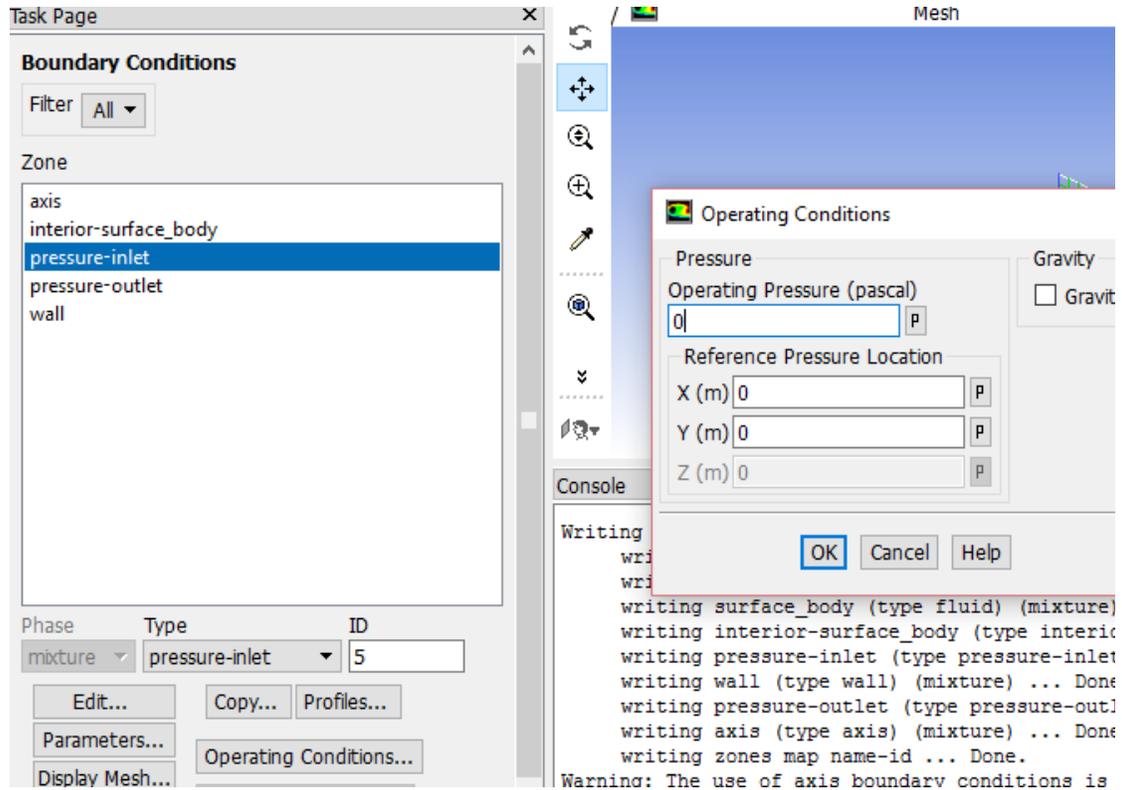
Select *air* under *Fluid materials*. Under Properties, choose *Ideal Gas* next to *Density*. You should see the window expand. This means FLUENT uses the ideal gas equation of state to relate density to the static pressure and temperature.



Click *Change/Create*. Close the window.

Define > Operating Conditions

We'll work in terms of absolute rather than gauge pressures in this example. So set Operating Pressure in the Pressure box to 0.



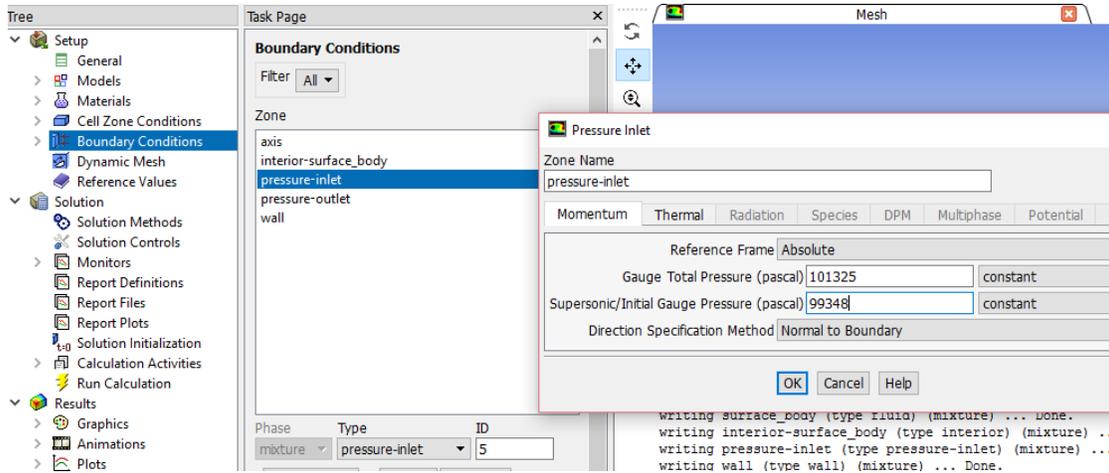
Click OK.

It is important that you set the operating pressure correctly in compressible flow calculations since FLUENT uses it to compute absolute pressure to use in the ideal gas law.

Define > Boundary Conditions

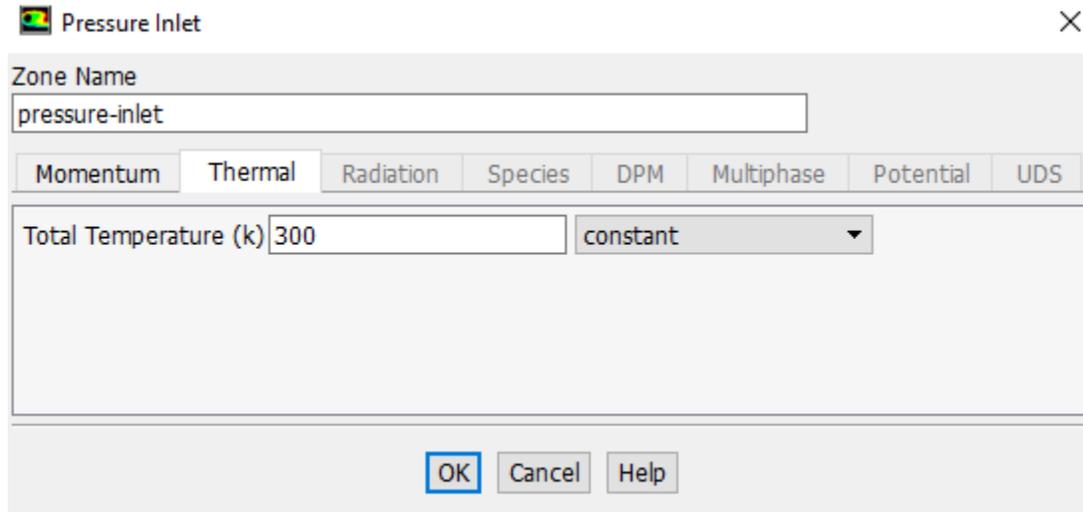
Set boundary conditions for the following surfaces: centerline, inlet, outlet, wall.

Select inlet under *Zone* and pick *pressure-inlet* under *Type* as its boundary condition. Click Set.... The *Pressure Inlet* window should come up.

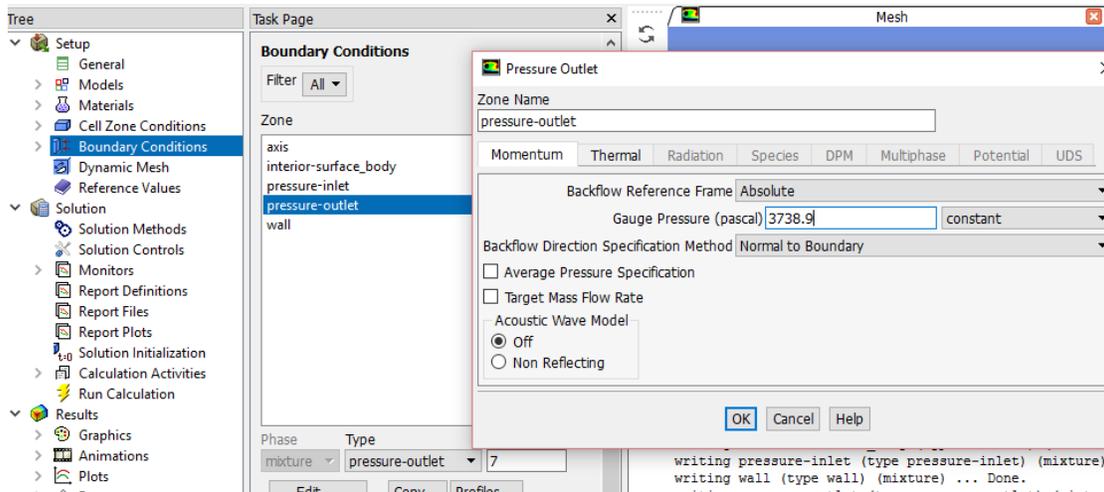


Set the total (i.e. stagnation) pressure (noted as Gauge Total Pressure in FLUENT) and temperature at the inlet. For a subsonic inlet, *Supersonic/Initial Gauge Pressure* is the initial guess value for the static pressure. Enter 99348 for the Supersonic/Initial Gauge Pressure. After you have entered the values, click OK to close the window.

Under the Thermal tab, the Total Temperature should be 300.



Using the same steps as above, pick *pressure-outlet* as the boundary condition for the *outlet* surface. Then, when the *Pressure Outlet* window comes up, set the total pressure to 3738.9 as specified in the problem statement.



Click OK.

Make sure that wall zone is set to wall type and the centerline zone is set to axis type.

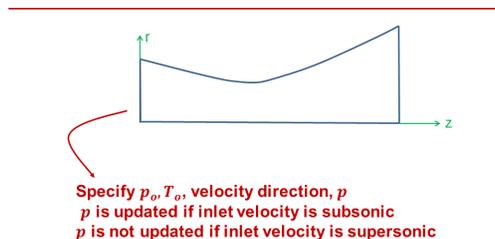
Set Reference Values

Under Setup > Click Reference values > from the compute from drop down menu > select pressure-inlet

These reference value numbers will be used in calculating non-dimensional number such as drag coefficient

The gauge total pressure at the inlet boundary is the total (stagnation) pressure. This value is taken as is. The supersonic/initial gauge pressure is an initial guess for the static pressure at the inlet. This value will get updated if the flow at the inlet is subsonic. This value will not get updated if the inlet velocity is supersonic since disturbances cannot propagate upstream in this case. In our problem, the inlet velocity is going to be subsonic, so the static pressure at the inlet will get updated. You just need to make sure you provide a reasonable initial guess for the static pressure so that the iterations converge. This is illustrated in the figure below.

“Pressure Inlet” Boundary Condition



Keep in mind that all pressures you enter are gauge: Absolute pressure = Gauge pressure + Reference pressure. The reference pressure is set to zero under "Operating conditions" in the tutorial video. So, there is no difference between gauge and absolute pressures. This is a good idea for highly compressible flows where one gets significant variation in pressure across the domain. In contrast, for low speed flows, where absolute pressure variation might not be significant, giving a reasonable reference pressure will ensure that gauge pressure variations are still significant and minimize rounds-off errors (essentially you avoid having to take small differences of large numbers).

Note: If using a different fluid than air, you must assign the material under the cell zone conditions. Do this by going to Cell Zone Conditions, clicking on 'surface body', selecting the material name that you want, and click OK.

Step 5: Numerical Solution

Watch the tutorial video.

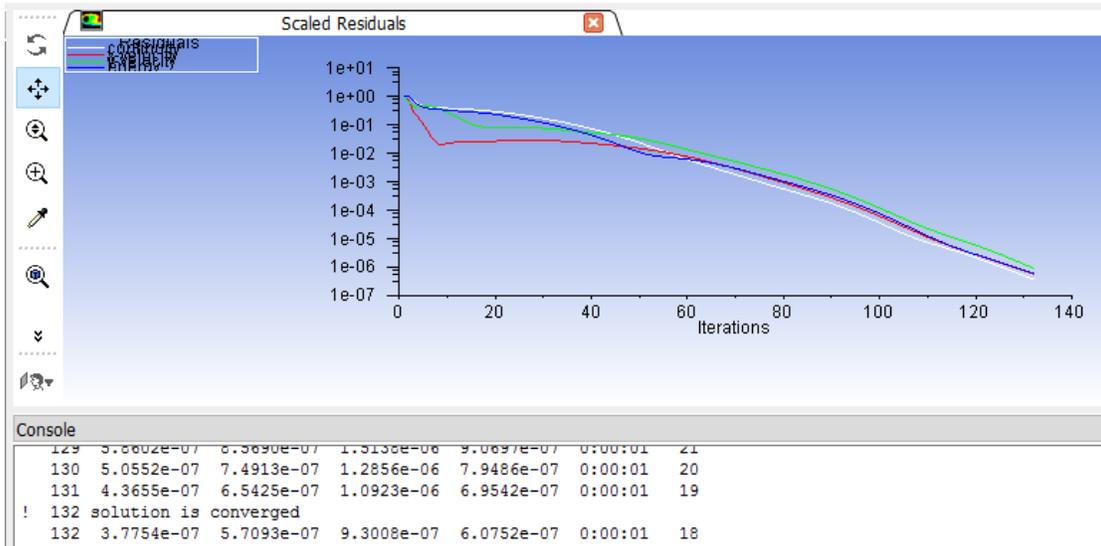
A summary of the video steps is shown below:

1. Under Solution Methods > Spatial Discretization
 - a. Set Gradient to Least Squares Cell Based
 - b. Set Flow to Second Order Upwing
2. Under Solution > Monitors
 - a. Residuals
 - i. Set all Residual Convergence Criteria to 1e-6
3. Under Solution > Solution Initialization
 - a. Select Standard Initialization > Compute from Inlet
4. Under Run Calculation
 - a. Set Number of Iterations to 150
 - b. Calculate

What does the convergence plot look like?

How many iterations does it take to converge?

Save case and data after you have obtained a converged solution.

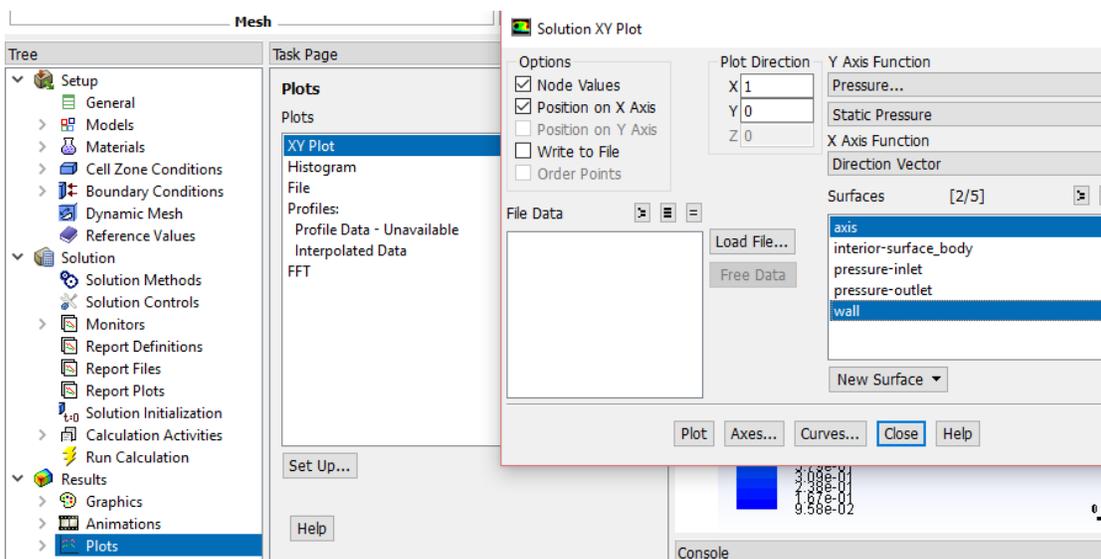


Step 6: Analyze Results

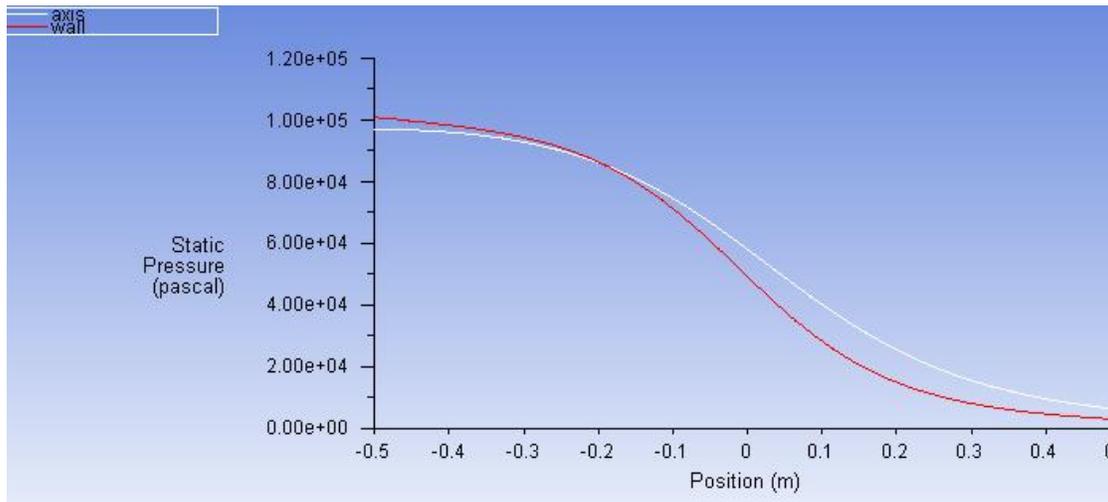
Static Pressure Plot

We are going to plot the velocity along the centerline and wall. We are going to use the Static Pressure.

Plot > XY Plot



Click Plot.



Mach Number Plot

We are going to plot the velocity along the centerline. However, this time, we are going to use the dimensionless Mach quantity.

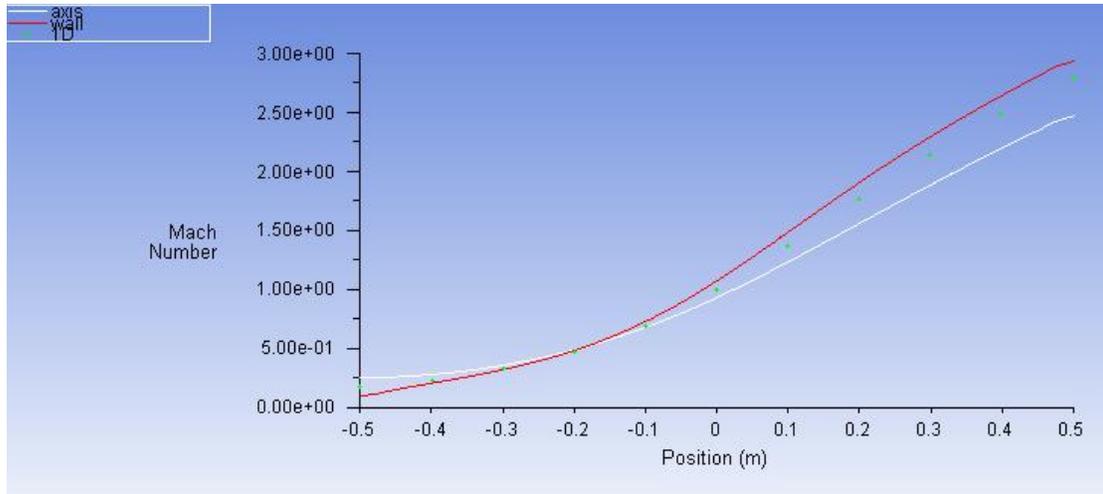
Plot > XY Plot

We are going to plot the variation of the Mach number in the axial direction at the axis and wall. In addition, we will plot the corresponding variation from 1D theory. You can download the file here: [mach_1D.xy](#).

Do everything as we would do for plotting the centerline velocity. However, instead of selecting Axial Velocity as the Y Axis Function, select Mach Number.

Also, since we are going to plot this number at both the wall and axis, select centerline and wall under Surfaces.

Then, load the [mach_1D.xy](#) by clicking on Load File....



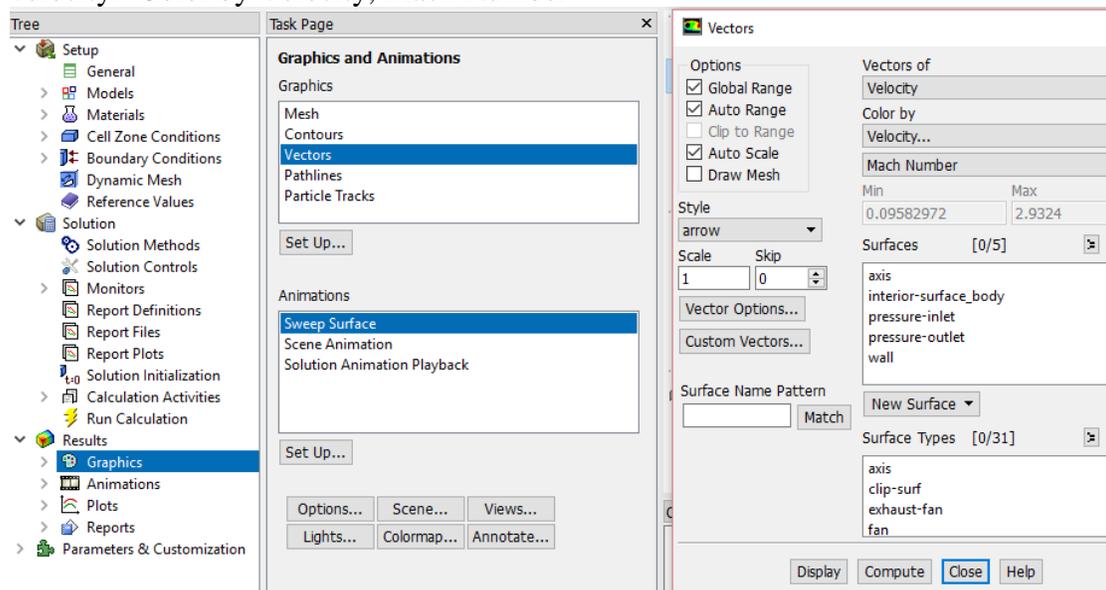
How does the FLUENT solution compare with the 1D solution?

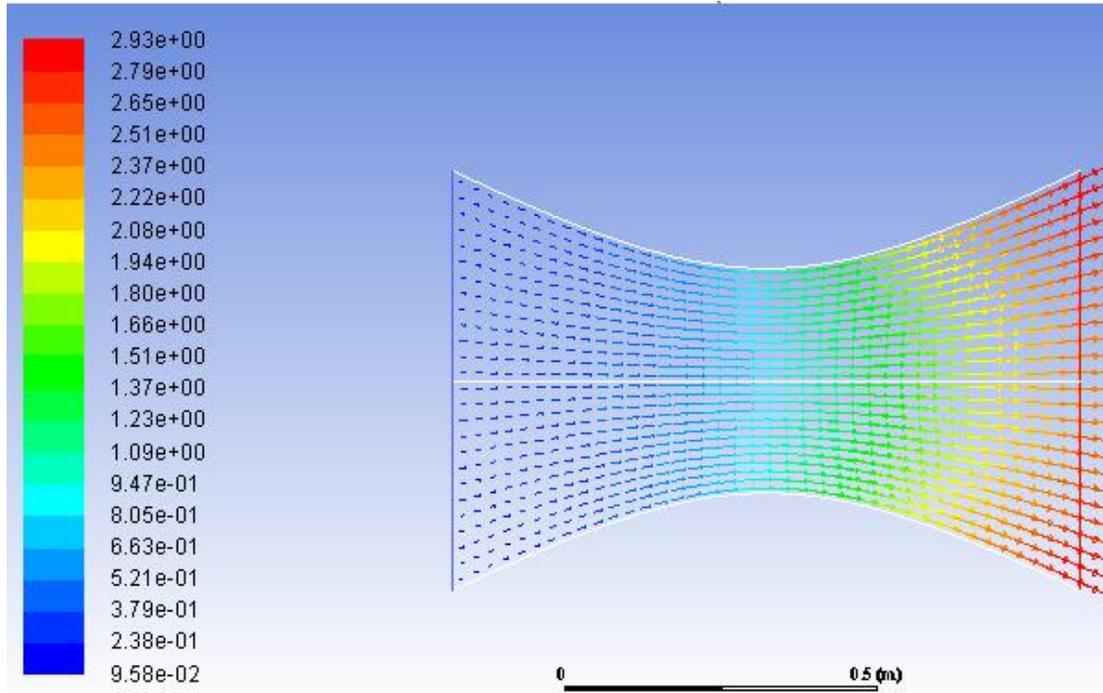
Is the comparison better at the wall or at the axis? Can you explain this?

Save this plot as machplot.xy by checking Write to File and clicking Write....

Velocity Vector Plot

Under Results > Graphics > Select Vectors> Press Setup > Choose vectors of velocity> Color by Velocity, Mach Number





Pressure Contour Plot

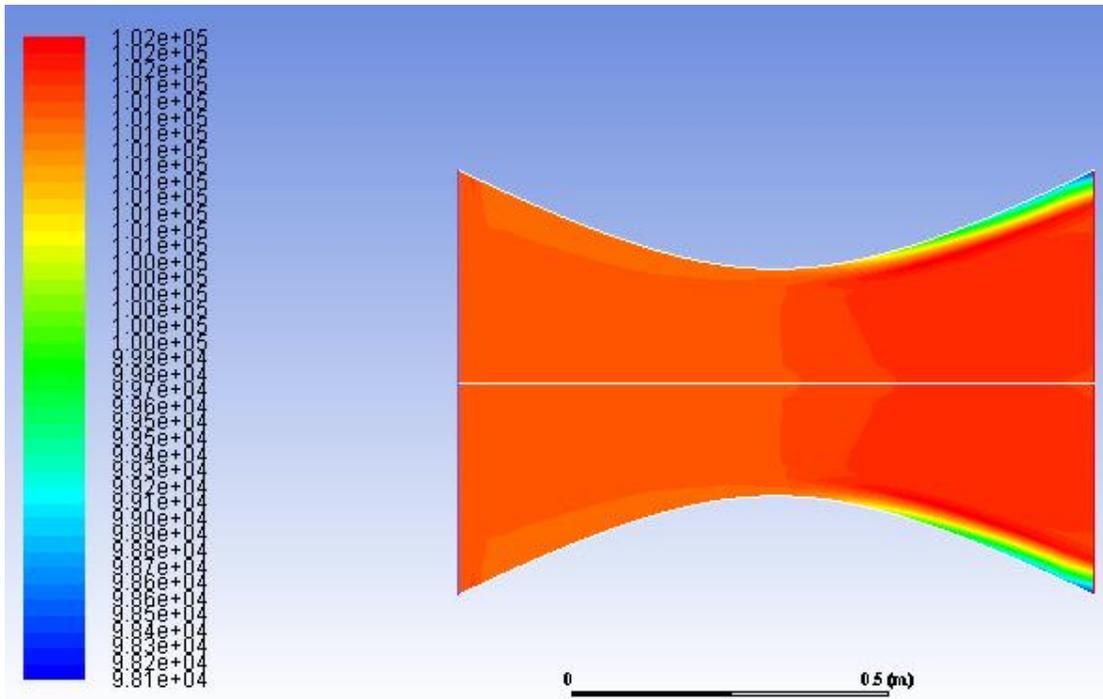
Sometimes, it is very useful to see how the pressure and temperature changes throughout the object. This can be done via contour plots.

Under Results > Graphics > Select Contours> Press Setup > Choose Contours of Pressure> Static Pressure

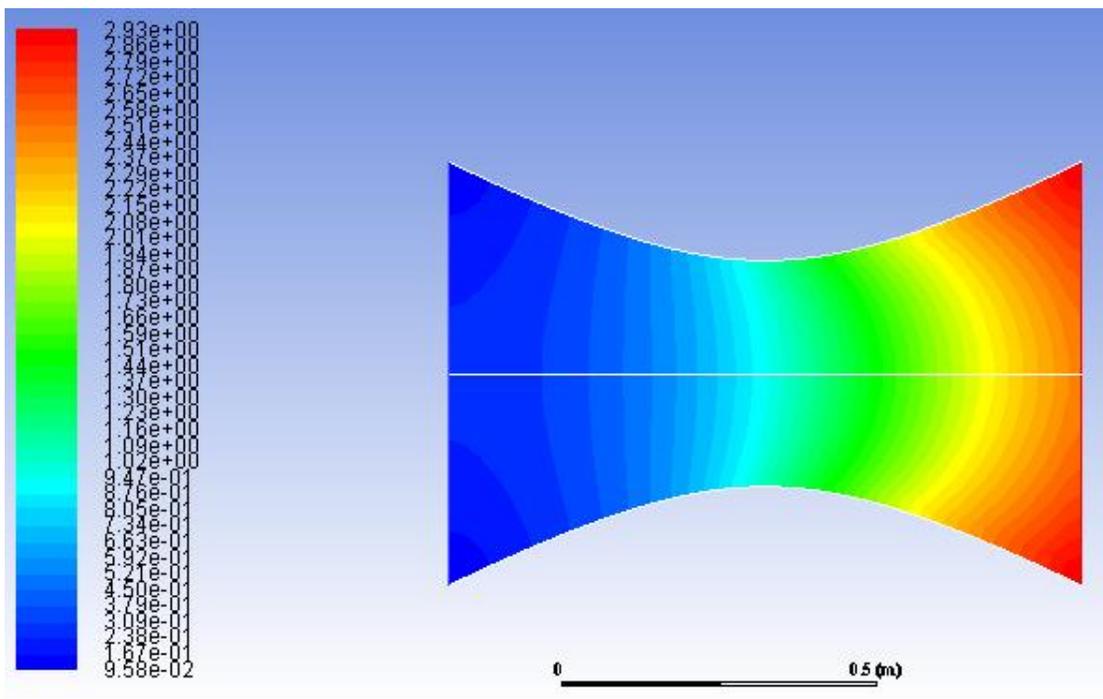
We want this at a fine enough granularity so that we can see the pressure changes clearly. Under Levels, change the default 20 to 40. This increases the number of lines in the contour plot so that we can get a more accurate result.

Notice that the pressure on the fluid gets smaller as it flows to the right, as is consistent with fluid going through a nozzle.

Total pressure Contour are displayed below



Mach Number Contour Plot

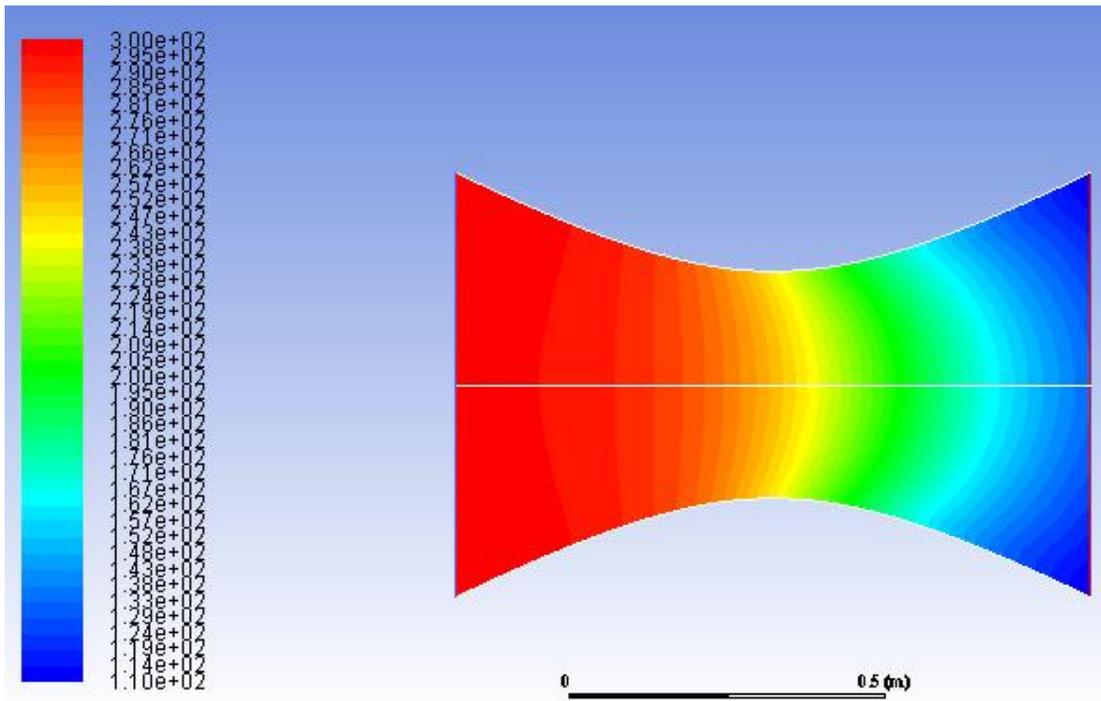


Temperature Contour Plot

Now we will plot the temperature contours and see how the temperature varies throughout the nozzle.

Back in the *Contours* window, under Contours Of, select Temperature... and Static Temperature.

Click Display.



As we can see, the temperature decreases towards the right side of the nozzle, indicating a change of internal energy to kinetic energy as the fluid speeds up.

Step 7: Refine Mesh

Solve the nozzle flow for the same conditions as before and change the element size to 0.0125 m instead of 0.025 m.

(a) Plot the variation of Mach number at the axis and the wall as a function of the axial distance x . Also, plot the corresponding results obtained on the first mesh and from the quasi-1D assumption. Recall that the quasi-1D result for the Mach number variation was given to you in the M_1D.xy file. Note all five curves should be plotted on the same graph so that you can compare them. You can make the plots in

FLUENT, MATLAB or EXCEL.

(b) Plot the variation of static pressure at the axis and the wall as a function of the axial distance x . Also, plot the corresponding results obtained on the 50x20 grid used in class and from the quasi-1D assumption.

(c) Plot the variation of static temperature at the axis and the wall as a function of the axial distance x . Also, plot the corresponding results obtained on the 50x20 grid used in class and from the quasi-1D assumption. Calculate the static temperature variation for the quasi-1D case.

Comment very briefly on the grid dependence of your results and the comparison with the quasi-1D results.