

FLUENT - Compressible Flow in a Nozzle- Step 4

Author: Rajesh Bhaskaran & Yong Sheng Khoo, Cornell University

[Problem Specification](#)

[1. Pre-Analysis & Start-up](#)

[2. Geometry](#)

[3. Mesh](#)

[4. Setup \(Physics\)](#)

[5. Solution](#)

[6. Results](#)

[7. Verification and Validation](#)

[Problem 1](#)

[Problem 2](#)

Step 4: Setup (Physics)



Useful Information

These instructions are for FLUENT 6.3.26. [Click here](#) for instructions for FLUENT 12.

If you have skipped the previous mesh generation steps 1-3, you can download the mesh by right-clicking on [this link](#). Save the file as *nozzle.msh* in your working directory. You can then proceed with the flow solution steps below.

Launch FLUENT

Start > Programs > Fluent Inc > FLUENT 6.3.26 > FLUENT 6.3.26

Select *2ddp* from the list of options and click *Run*.

Import File

Main Menu > File > Read > Case...

Navigate to your working directory and select the *nozzle.msh* file. Click *OK*.

The following should appear in the FLUENT window:

```
> Reading "H:\FLUENT Tutorials\nozzle\nozzle.msh"...
1071 nodes.
  50 mixed wall faces, zone 3.
  50 mixed wall faces, zone 4.
  20 mixed wall faces, zone 5.
  20 mixed wall faces, zone 6.
1930 mixed interior faces, zone 8.
1000 quadrilateral cells, zone 2.

Building...
  grid,
  materials,
  interface,
  domains,
  zones,
  default-interior
  inlet
  outlet
  centerline
  wall
  fluid
  shell conduction zones,
Done.
```

Check that the displayed information is consistent with our expectations of the nozzle grid.

Check and Display Grid

First, we check the grid to make sure that there are no errors.

Main Menu > Grid > Check

Any errors in the grid would be reported at this time. Check the output and make sure that there are no errors reported.

Grid > Info > Size

How many cells and nodes does the grid have?

Main Menu > Display > Grid

Make sure all items under **Surfaces** is selected. Then click **Display**. The graphics window opens and the grid is displayed in it.

Some of the operations available in the graphics window are:

Translation: The grid can be translated in any direction by holding down the **Left Mouse Button** and then moving the mouse in the desired direction.

Zoom In: Hold down the **Middle Mouse Button** and drag a box from the **Upper Left Hand Corner** to the **Lower Right Hand Corner** over the area you want to zoom in on.

Zoom Out: Hold down the **Middle Mouse Button** and drag a box anywhere from the **Lower Right Hand Corner** to the **Upper Left Hand Corner**.

The grid has 50 divisions in the axial direction and 20 divisions in the radial direction. The total number of cells is $50 \times 20 = 1000$. Since we are assuming inviscid flow, we won't be resolving the viscous boundary layer adjacent to the wall. (The effect of the boundary layer is small in our case and can be neglected.) Thus, we don't need to cluster nodes towards the wall. So the grid has uniform spacing in the radial direction. We also use uniform spacing in the axial direction.

Look at specific parts of the grid by choosing each boundary (centerline, inlet, etc) listed under **Surfaces** in the *Grid Display menu*. Click to select and click again to deselect a specific boundary. Click **Display** after you have selected your boundaries.

Define Solver Properties

Define > Models > Solver...

We see that FLUENT offers two methods ("solvers") for solving the governing equations: Pressure-Based and Density-Based. To figure out the basic difference between these two solvers, let's turn to the documentation.

Main Menu > Help > User's Guide Contents ...

This should bring up *FLUENT 6.3 User's Guide* in your web browser. If not, access the User's Guide from the Start menu: **Start > Programs > Fluent Inc Products > Fluent 6.3 Documentation > Fluent 6.3 Documentation**. This will bring up the FLUENT documentation in your browser. Click on the link to the user's guide.

Go to chapter 25 in the user's guide; it discusses the Pressure-Based and Density-Based solvers. Section 25.1 introduces the two solvers:

"Historically speaking, the pressure-based approach was developed for low-speed incompressible flows, while the density-based approach was mainly used for high-speed compressible flows. However, recently both methods have been extended and reformulated to solve and operate for a wide range of flow conditions beyond their traditional or original intent."

"In both methods the velocity field is obtained from the momentum equations. In the density-based approach, the continuity equation is used to obtain the density field while the pressure field is determined from the equation of state."

"On the other hand, in the pressure-based approach, the pressure field is extracted by solving a pressure or pressure correction equation which is obtained by manipulating continuity and momentum equations."

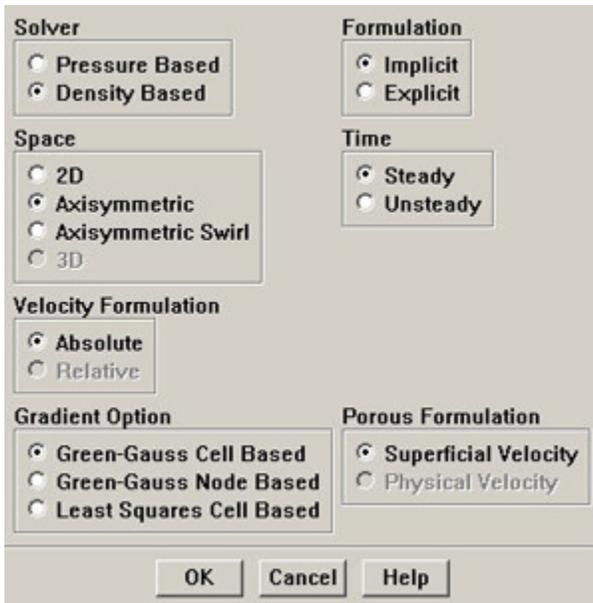
Mull over this and the rest of this section. So which solver do we use for our nozzle problem? Turn to section 25.7.1 in chapter 25:

"The pressure-based solver traditionally has been used for incompressible and mildly compressible flows. The density-based approach, on the other hand, was originally designed for high-speed compressible flows. Both approaches are now applicable to a broad range of flows (from incompressible to highly compressible), but the origins of the density-based formulation may give it an accuracy (i.e. shock resolution) advantage over the pressure-based solver for high-speed compressible flows."

Since we are solving a high-speed compressible flow, let's pick the density-based solver.

In the **Solver** menu, select **Density Based**.

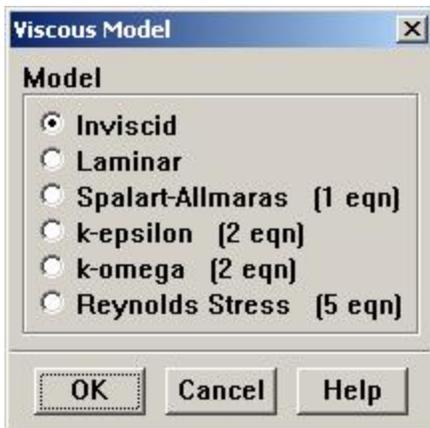
Under **Space**, choose **Axisymmetric**. This will solve the axisymmetric form of the governing equations.



Click **OK**.

Define > Models > Viscous

Select **Inviscid** under **Model**.

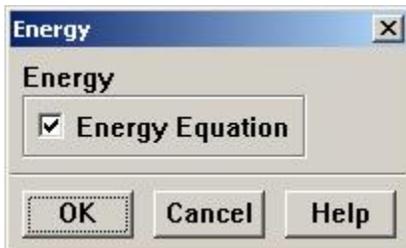


Click **OK**. This means the solver will neglect the viscous terms in the governing equations.

Define > Models > Energy

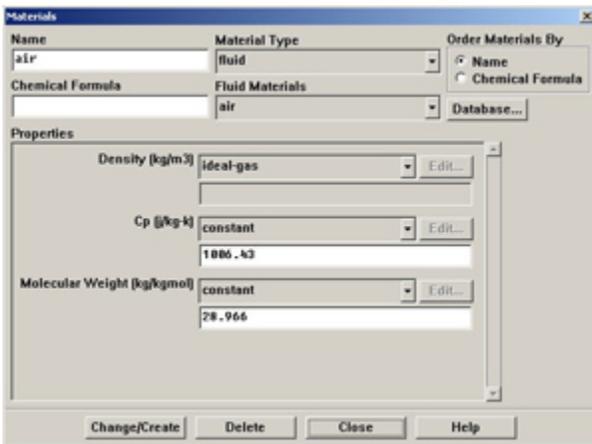
The energy equation needs to be turned on since this is a compressible flow where the energy equation is coupled to the continuity and momentum equations.

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



Define > Materials

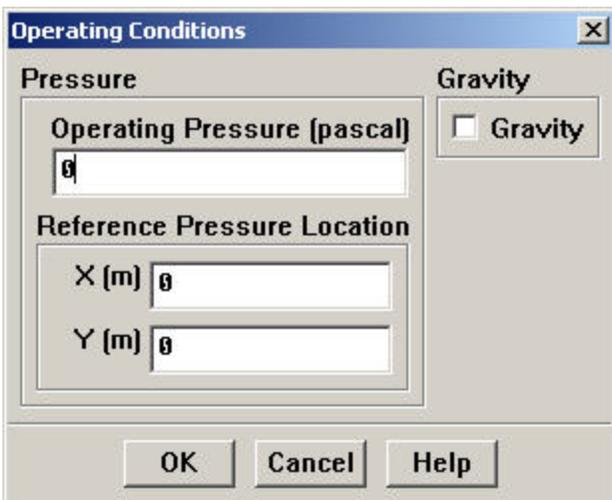
Select **air** under **Fluid materials**. Under **Properties**, choose **Ideal Gas** next to **Density**. You should see the window expand. This means FLUENT uses the ideal gas equation of state to relate density to the static pressure and temperature.



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**.

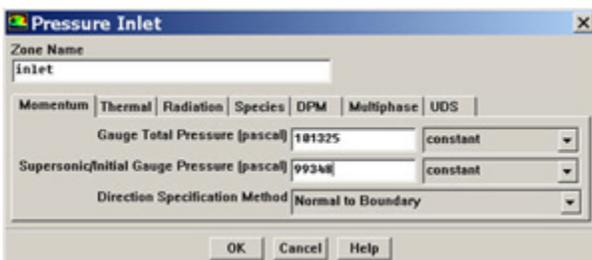
It is important that you set the operating pressure correctly in compressible flow calculations since FLUENT uses it to compute absolute pressure to use in the ideal gas law.

Define > Boundary Conditions

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

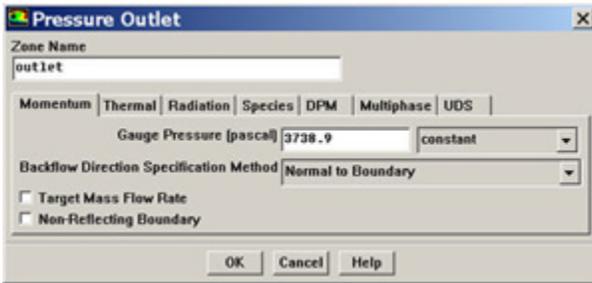
Select **inlet** under **Zone** and pick **pressure-inlet** under **Type** as its boundary condition. Click **Set...**. The **Pressure Inlet** window should come up.

Set the total pressure (noted as **Gauge Total Pressure** in FLUENT) at the inlet to 101,325 Pa as specified in the problem statement. For a subsonic inlet, **Supersonic/Initial Gauge Pressure** is the initial guess value for the static pressure. This initial guess value can be calculated from the 1D analysis since we know the area ratio at the inlet. This value is 99,348 Pa. Note that this value will be updated by the code. After you have entered the values, click **OK** to close the window.



Check that under the **Thermal** tab, the **Total Temperature** is 300 K. Click **OK**.

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 pressure to 3738.9 as specified in the problem statement. Click *OK*.



Set the *centerline* zone to *axis* boundary type.

Make sure that *wall* zone is set to *wall* boundary type.

Go to [Step 5: Solution](#)

[See and rate the complete Learning Module](#)

[Go to all FLUENT Learning Modules](#)