

# FLUENT - Supersonic Flow Over a Wedge - Step 4 \*New

## Problem Specification

1. Pre-Analysis & Start-up
2. Geometry
3. Mesh
- 4. Setup (Physics)**
5. Solution
6. Results
7. Verification & Validation



## Site Under Construction

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



## Useful Information

Click on the images to display a higher resolution image on your browser.

## Step 4: Set Up Problem in FLUENT

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 *wedge.msh*. You can then proceed with the flow solution steps below.

### Launch FLUENT

**Start > All Programs > ANSYS 12.0 > Fluid Dynamics > FLUENT**

Select **2D** under the Dimension list and **Double Precision** under the Options list, and click **Run**.

In the double-precision solver, each floating point number is represented using 64 bits in contrast to the single-precision solver which uses 32 bits. The extra bits increase not only the precision but also the range of magnitudes that can be represented. The downside of using double precision is that it requires more memory.

### Import File

**File > Read > Mesh...**

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

Check that the displayed information is consistent with our expectations.

### Analyze Grid

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

**Problem Setup > General > Check**

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

**Mesh > Info > Size**

How many cells and nodes does the grid have?

**General > Mesh > Display...**

You can look at specific parts of the grid by choosing the items you wish to view under **Surfaces** (click to select and click again to deselect a specific boundary). Click **Display** again when you have selected your boundaries. Note what the surfaces *farfield*, *wedge*, etc. correspond to by selecting and plotting them in turn.

### Define Properties

## General > Solver > Type

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.

### Help > User's Guide Contents ...

This should bring up *ANSYS FLUENT 12.0 User's Guide* in your web browser. If not, access the User's Guide from the Start menu: **Start > Programs > ANSYS 12.0 > Help > FLUENT Help**. This will bring up the FLUENT documentation in your browser. Click on the link to the user's guide.

Go to chapter 26 in the user's guide; it discusses the Pressure-Based and Density-Based solvers. Section 26.1 introduces the two solvers:

*(The following is not the exact text from the help file.)*

"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."

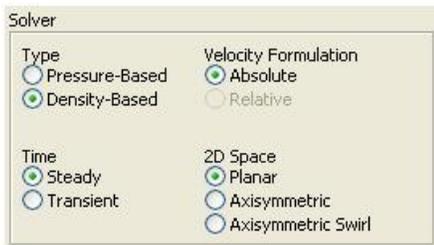
"In both methods the velocity field is obtained from the momentum equations. In the density-based approach, the continuity equation is used to obtain the density field while the pressure field is determined from the equation of state."

"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 wedge 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 expect an oblique shock for our problem and the density-based solver is likely to resolve the shock better, let's pick this solver.

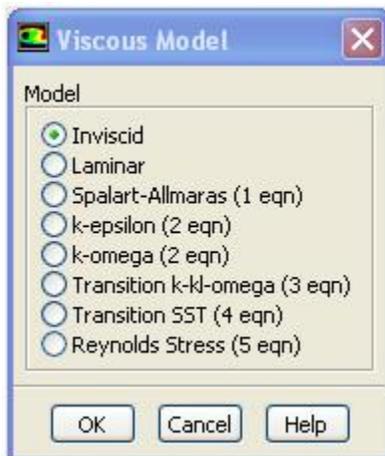


In the **Solver** menu, select **Density-Based**.

### Problem Setup > Models

Double click on **Viscous - Laminar**

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



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

### Models > Energy

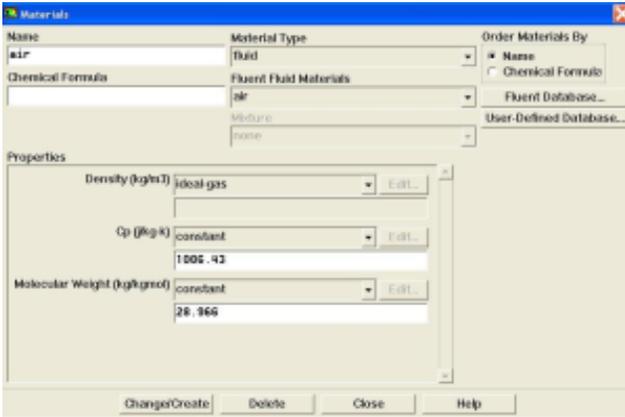
Double click on **Energy - Off**

In compressible flow, the energy equation is coupled to the continuity and momentum equations. So we need to solve the energy equation for our problem.

To turn on the energy equation, check the box next to **Energy Equation** and click **OK**.

### Problem Setup > Materials

Make sure air is selected under **Fluid**. Double click air, set **Density** to **ideal-gas** and make sure **Cp** is **constant** and equal to 1006.43 j/kg-k. Also make sure the Molecular Weight is constant and equal to 28.966 kg /kgmol. Selecting the **ideal gas** option means that FLUENT will use the ideal-gas equation of state to relate density to the static pressure and temperature.



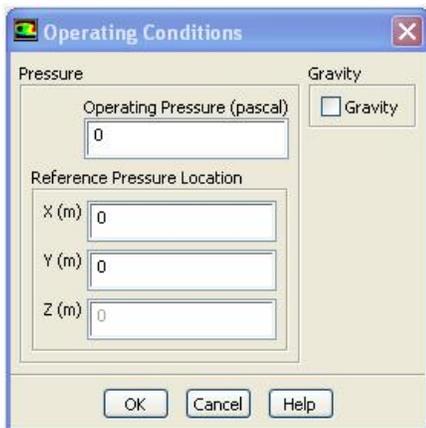
Click **Change/Create**.

### Define > Operating Conditions

To understand what the **Operating Pressure** is, read through the short-and-sweet section 8.14.2 in the user's guide. We see that for all flows, FLUENT uses the gauge pressure internally in order to minimize round-off errors. Any time an absolute pressure is needed, as in the ideal gas law, it is generated by adding the **operating pressure** to the gauge pressure:

$$\text{absolute pressure} = \text{gauge pressure} + \text{operating pressure}$$

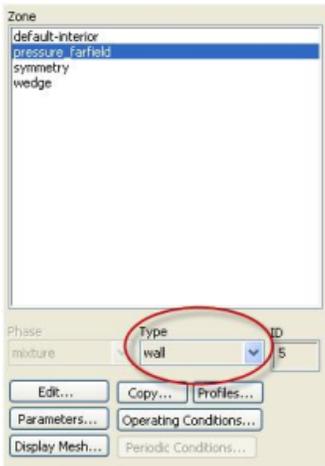
Round-off errors occur when pressure changes  $p$  in the flow are much smaller than the pressure values  $p$ . One then gets small differences of large numbers. For our supersonic flow, we'll get significant variation in the absolute pressure so that pressure changes  $p$  are comparable to pressure levels  $p$ . So we can work in terms of absolute pressure without being hassled by pesky round-off errors. To have FLUENT work in terms of the absolute pressure, set the **Operating Pressure** to 0.



Thus, in our case, there is no difference between the gauge and absolute pressures. Click **OK**.

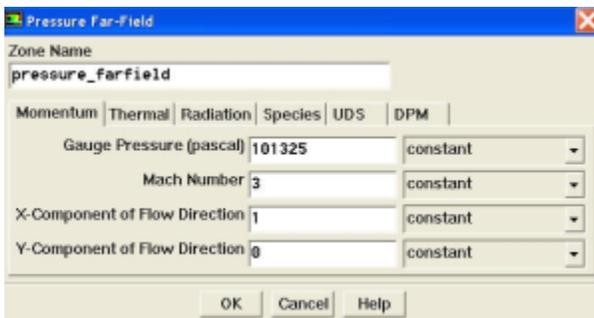
### Define > Boundary Conditions

Set the boundary condition for the *pressure\_farfield* surface (aka zone) to the boundary type **pressure-far-field** by clicking on the drop-down list. Select **Yes** in the pop-up window asking if it's "OK to change pressure\_farfield's type from wall to pressure-far-field?".



Set the **Gauge Pressure** to 101325. Set the **Mach Number** to 3. Under **X-Component of Flow Direction**, enter a value of 1 (i.e. the farfield flow is in the X direction).

Next, click on the **Thermal** Tab. Change the temperature to 300K.



Click **OK**. The **pressure-far-field** boundary type effectively imposes that there is no upstream propagation of disturbances if the flow at the boundary is supersonic. See section 7.9 of the FLUENT help for more details about this boundary type.

Similarly, change the boundary condition for the **symmetry** surface to the **symmetry** boundary type. No user input is required for the **symmetry** boundary type. At any boundary set to the **symmetry** type, FLUENT internally sets

- normal velocity = 0
- normal gradients of all variables = 0

See section 7.14 of the FLUENT help for more details.

The boundary type for the **wedge** surface is set to **wall** by default. There is no need to change that.

Go to [Step 5: Solution!](#)

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

Go to [all FLUENT Learning Modules](#)