FLUENT - Supersonic Flow Over a Wedge- Step 4 (FLUENT 6.3.26)

ⓘ **Useful Information**

Click Here for the FLUENT 12 version.

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 > Programs > Fluent Inc > FLUENT 6.3.26

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

The "2ddp" option is used to select the two-dimensional (2d), double-precision (dp) solver. 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

Main Menu > File > Read > Case...

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.

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?

Display > Grid

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

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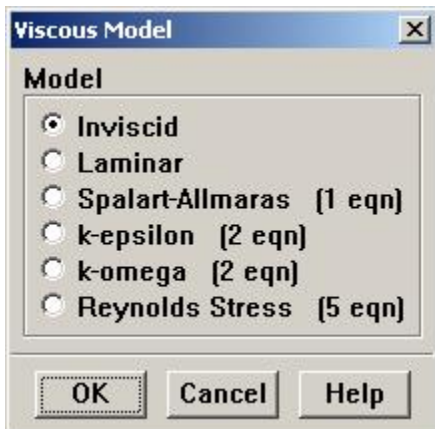
In the *Solver* menu, select *Density Based*.
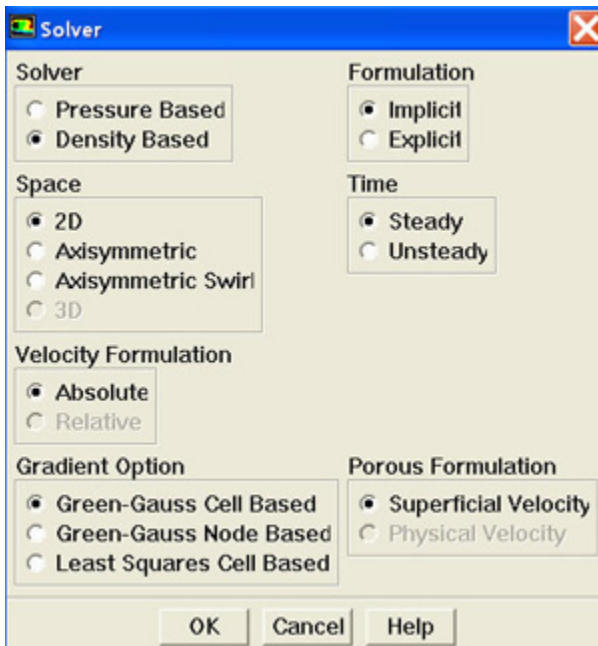
Click *OK*.

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

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*.

Define > Materials

Make sure air is selected under *Fluid Materials*. 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.
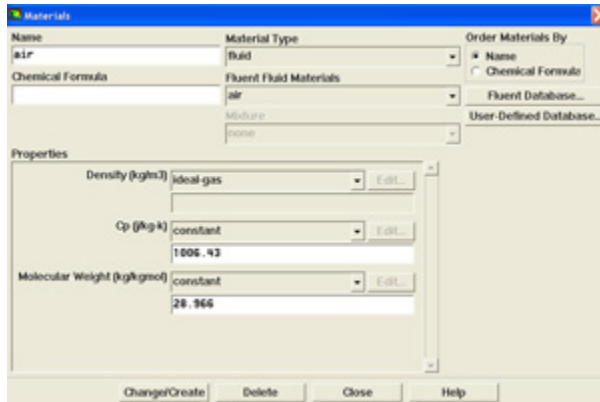

Higher Resolution Image

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:

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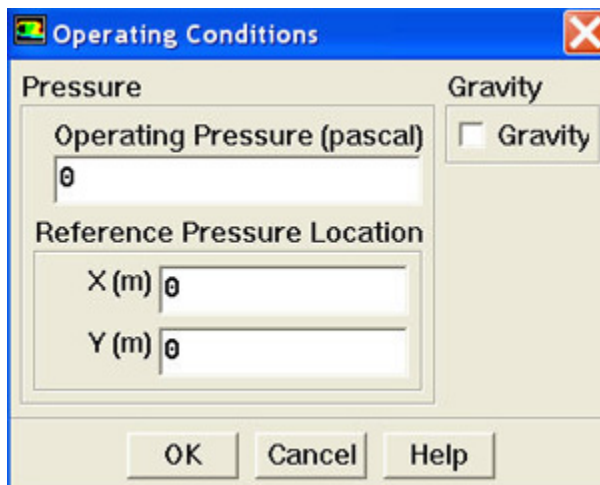
Thus, in our case, there is no difference between the gauge and absolute pressures. Click *OK*.

Define > Boundary Conditions

Set the boundary condition for the *pressure_farfield* surface (aka zone) to the *pressure-far-field* boundary type by clicking on the latter. Select *Yes* in the pop-up window asking if it's "OK to change pressure_farfield's type from wall to pressure-far-field?".
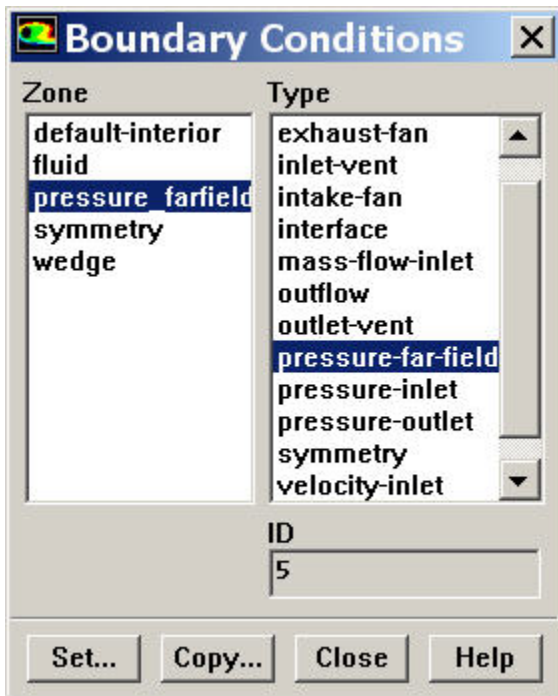
Set the **Gauge Pressure** to 101325. Set the **Mach Number** to 3. Under **X-Component of Flow Direction**, enter a value of 1 (i.e. the farfield flow is in the X direction).

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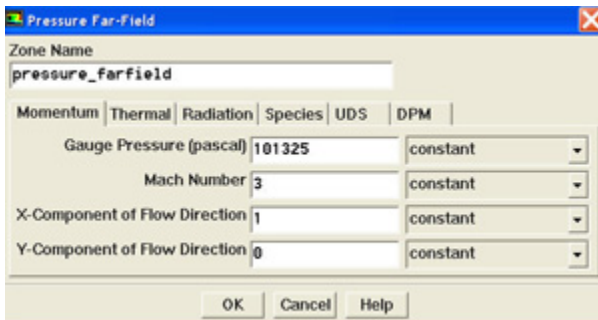
Higher Resolution Image

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: Solve!

See and rate the complete Learning Module

Go to all FLUENT Learning Modules