Supersonic Flow Over a Wedge - Physics Setup

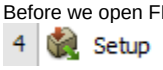
Author: Rajesh Bhaskaran, Cornell University

Physics Setup

Note that the look and arrangement of the menu setup in FLUENT has changed a bit since this tutorial was created. If something has changed in the menu setup from the description below, you can often find the equivalent options by poking around in the menus.

Update the Project and Open FLUENT

Before we open FLUENT, we need to right-click on the Mesh cell and choose *Update*. You should see a checkmark next to Mesh. Double-click Setup

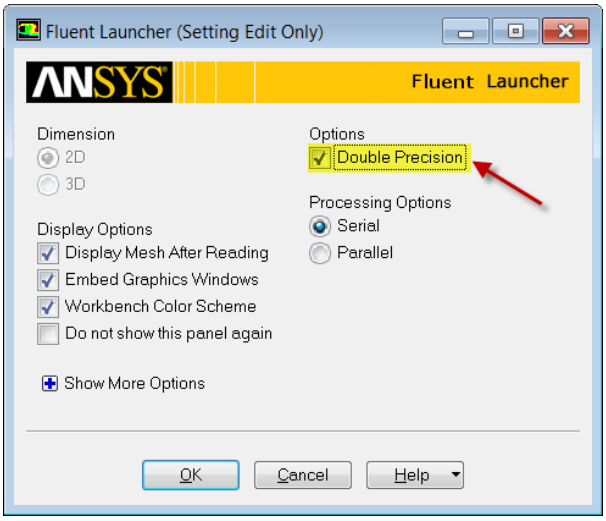


to open FLUENT.

Initial Settings

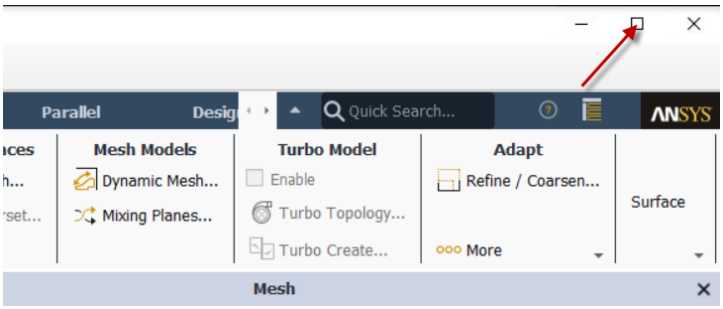
Double-Click Setup in the ***Workbench Project Page***.

When the ***FLUENT Launcher*** appears, choose "Double Precision" under "Options" and then click ***OK*** as shown below.The Double Precision option is used to select the double-precision 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.



Problem Setup - General

Now, FLUENT should open. If you are running FLUENT on Apps on Demand, you may need to hit the Maximize button (highlighted with the arrow in the snapshot below) to scroll down the tree in the Outline View.

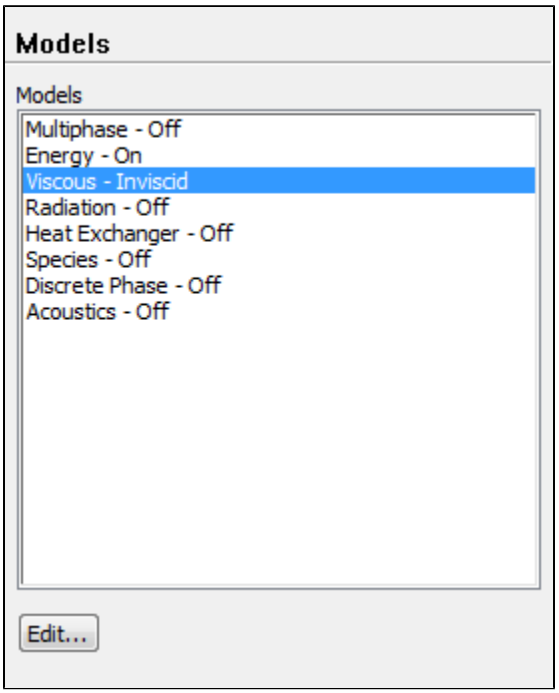


We will begin setting up some options for the solver. In the left hand window (the *Outline* window), under ***Problem Setup***, select ***General*** by double clicking. The only option we need to change here is the type of solver. In the *Solver* window, select ***Density-Based***. It is also a good idea to click the "Check" button under the Mesh options, as this will check if the mesh was created and transferred correctly.

Models

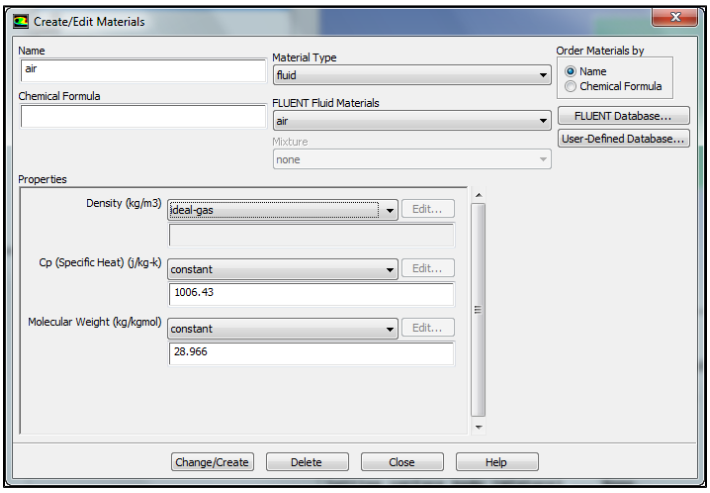
In the outline window, double-click ***Models***. We will need to utilize the energy equation in order to solve this simulation. Under *Models* highlight ***Energy - Off*** and click ***Edit...***. Now, the *Energy* window will launch. Check the box next to ***Energy Equation*** and hit OK. Doing this turns on the energy equation.

We also need to change the type of viscosity model. Select ***Viscous - Laminar*** and click ***Edit...***. Choose the ***Inviscid*** option and press ***OK***.



Materials

In the *Outline* window, double-click ***Materials***. In the *Materials* window, highlight ***Fluid > Air***, and click ***Create/Edit...***. this will launch the *Create/Edit Materials* window; here we can specify the properties of the fluid. Set the ***Density*** to ***Ideal Gas***, the ***Specific Heat*** to `1006.43`, and the ***Molecular Weight*** to `28.966`. When you have updated these fields, press ***Change/Create***. Then click *Close*.

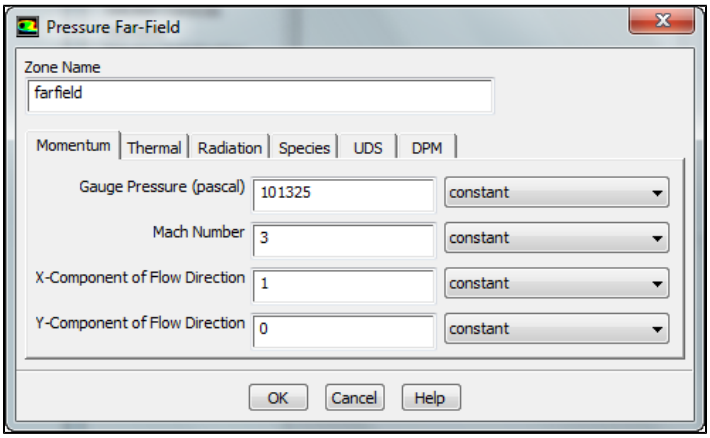


Boundary Conditions

In the *Outline* window, double-click ***Boundary Conditions***. We will now specify each boundary condition for the simulation.

Farfield

In the *Boundary Conditions* window, select ***farfield***. Use the drop-down menu to change the ***Type*** to ***pressure-far-field***. You will be asked to confirm the change, and do so by pressing ***OK***. (Note: In newer versions, this step is not needed.) Next, a dialogue box will open with some parameters we need to specify. Change the ***Gauge Pressure (Pascal)*** to ***101325***, and ***Mach Number*** to ***3***.



Also, select the ***Thermal*** tab, and ensure that the temperature correctly defaulted to ***300 K***. When you are finished, press ***OK***.

Wedge

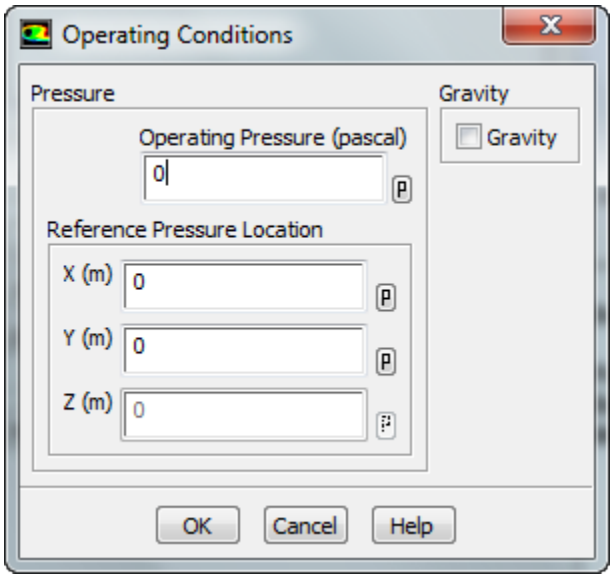
In the *Boundary Conditions* window, select ***wedge***. Use the drop-down menu to change the ***Type*** to ***wall***.

Symmetry

In the *Boundary Conditions* window, select ***symmetry***. Use the drop-down menu to change the ***Type*** to ***symmetry***.

Operating Conditions

In the *Boundary Conditions* window, select the ***Operating Conditions*** button. Change the ***Gauge Pressure*** to ***0***. Then press ***OK***



It is important to check the operating conditions. When setting the density in materials to *Ideal Gas*, FLUENT calculates the density using the absolute pressure. However, the pressure we specify is the gauge pressure, not the absolute pressure. FLUENT will use the absolute pressure to compute the density, therefore if we do not set the operating pressure to 0 our density will be incorrect for the flow field.

Go to Step 5: Numerical Solution

Go to all FLUENT Learning Modules