FLUENT - Flow over an Airfoil- Step 4

Step 4: Set Up Problem in FLUENT

Launch FLUENT

Start > Programs > Fluent Inc > FLUENT 6.3.26

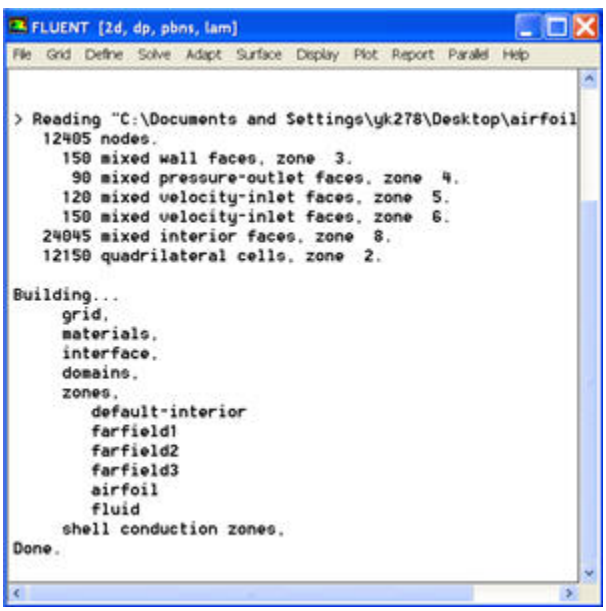
Select ***2ddp*** from the list of options and click ***Run***.

Import File

Main Menu > File > Read > Case...

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

The following should appear in the FLUENT window:



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

Analyze Grid

Grid > Info > Size

How many cells and nodes does the grid have?

Display > Grid

Note what the surfaces *farfield1, farfield2*, etc. correspond to by selecting and plotting them in turn.

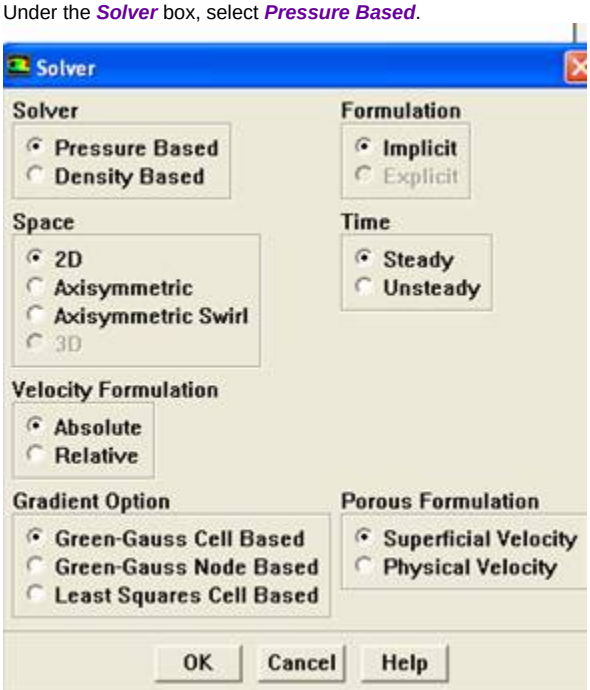
Zoom into the airfoil.

Where are the nodes clustered? Why?

Define Properties

Define > Models > Solver...

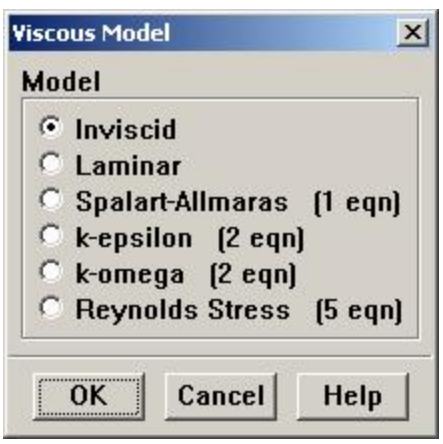
Under the ***Solver*** box, select ***Pressure Based***.



Click ***OK***.

Define > Models > Viscous

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



Click ***OK***.

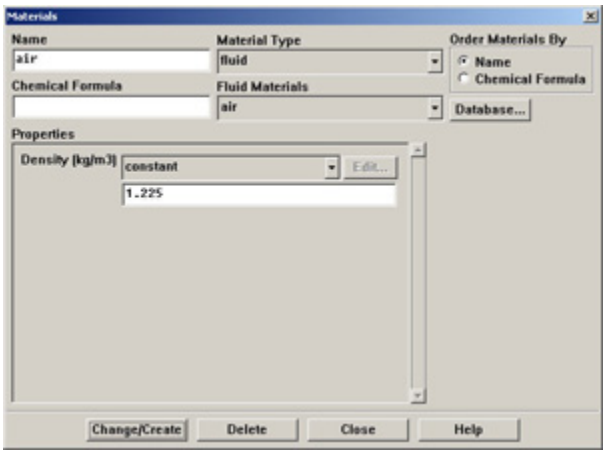
Define > Models > Energy

The speed of sound under SSL conditions is 340 m/s so that our freestream Mach number is around 0.15. This is low enough that we'll assume that the flow is incompressible. So the energy equation can be turned off.

Make sure there is no check in the box next to ***Energy Equation*** and click ***OK***.

Define > Materials

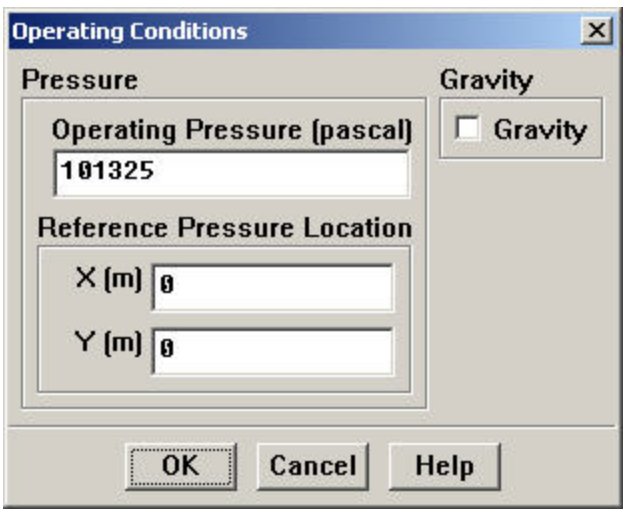
Make sure ***air*** is selected under ***Fluid Materials***. Set ***Density*** to ***constant*** and equal to 1.225 kg/m^3.



Click ***Change/Create***.

Define > Operating Conditions

We'll work in terms of gauge pressures in this example. So set ***Operating Pressure*** to the ambient value of 101,325 Pa.

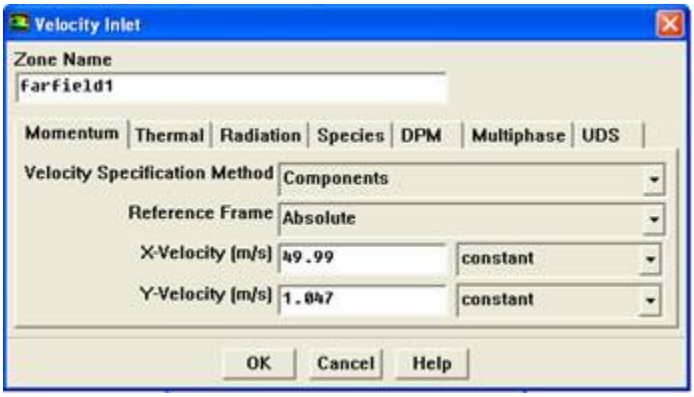


Click ***OK***.

Define > Boundary Conditions

Set ***farfield1*** and ***farfield2*** to the ***velocity-inlet*** boundary type.

For each, click Set.... Then, choose ***Components*** under ***Velocity Specification Method*** and set the x- and y-components to that for the freestream. For instance, the x-component is 50*cos(1.2)=49.99. (Note that 1.2° is used as our angle of attack instead of 2° to adjust for the error caused by assuming the airfoil to be 2D instead of 3D.)



Click ***OK***.

Set ***farfield3*** to ***pressure-outlet*** boundary type, click ***Set...*** and set the ***Gauge Pressure*** at this boundary to 0. Click ***OK***.

Go to Step 5: Solve!

See and rate the complete Learning Module

Go to all FLUENT Learning Modules