FLUENT - Turbulent Flow Past a Sphere - Step 4

UNDER CONSTRUCTION

Author: Daniel Kantor and Andrew Einstein, Cornell University

Step 4: Set Up Problem in FLUENT

Launch Fluent

Lab Apps > FLUENT 6.3.26

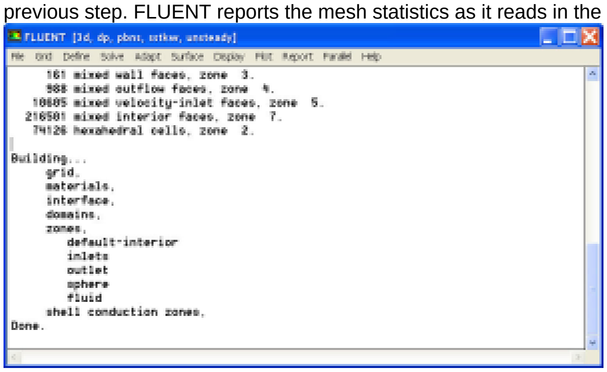
Select ***3ddp*** from the list of options and click ***Run***.

The "3ddp" option is used to select the 3-dimensional, 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.

Import Grid

Main Menu > File > Read > Case...

Navigate to the working directory and select the SingleSphere.msh file. This is the mesh file that was created using the preprocessor *GAMBIT* in the previous step. FLUENT reports the mesh statistics as it reads in the mesh:



Check the number of nodes, faces (of different types) and cells. Also, take a look under zones. We can see the three zones ***inlets***, ***outlet***, and ***sphere*** that we defined in *GAMBIT*.

Check and Display Grid

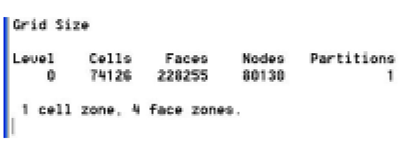
First, we check the grid to make sure that there are no errors.

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. Check the grid size:

Main Menu > Grid > Info > Size

The following info should appear (your number of cells might be slightly different because of slight different mesh criteria used):

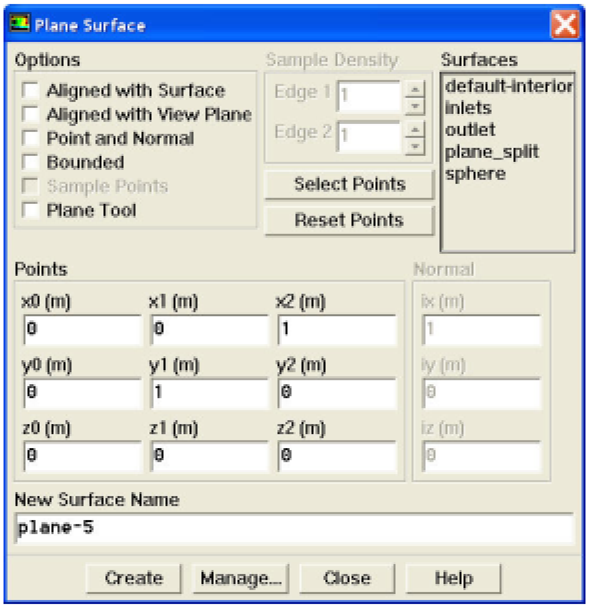


To view the grid we first need to create a plane that cuts our 3D model in half (otherwise it would be too hard to see a good profile of the mesh). To do this we go into:

Main Menu > Surfaces > Plane...

We change the values to:
x0 =0; x1=0; x2=1;
y0=0; y1=1; y2=0;
z0=0; z1=0; z2=0;

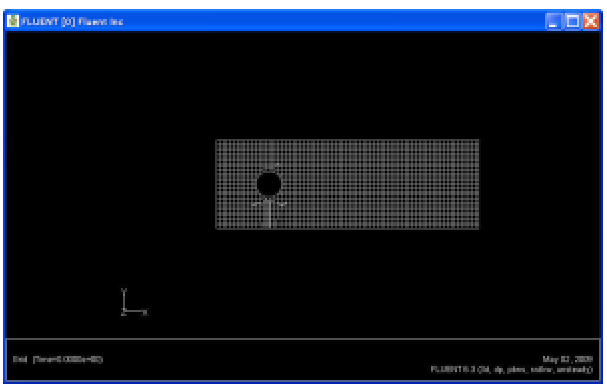
It will look like this:



We name this Plane_Split. To view the grid we now go to:

Main Menu > Display > Grid...

Make sure we select our surface under ***Surfaces***. Then click ***Display***. The graphics window opens and the grid is displayed in it. You can now click ***Close*** in the ***Grid Display*** menu to get back some desktop space. The graphics window will remain.



Graphics Window Operation

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***.

Use these operations to zoom into the grid to obtain the view shown below.

The zooming operations can only be performed with a middle mouse button.

White Background on Graphics Window

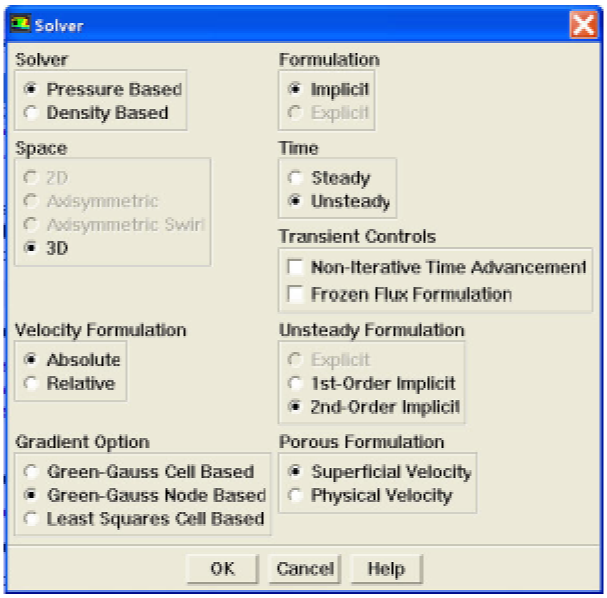
To get white background go to:
Main Menu > File > Hardcopy
Make sure that ***Reverse Foreground/Background*** is checked and select ***Color*** in ***Coloring*** section. Click ***Preview***. Click ***No*** when prompted "*Reset graphics window?*"

You can also look at specific parts of the grid by choosing the boundaries 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.

Define Solver Properties

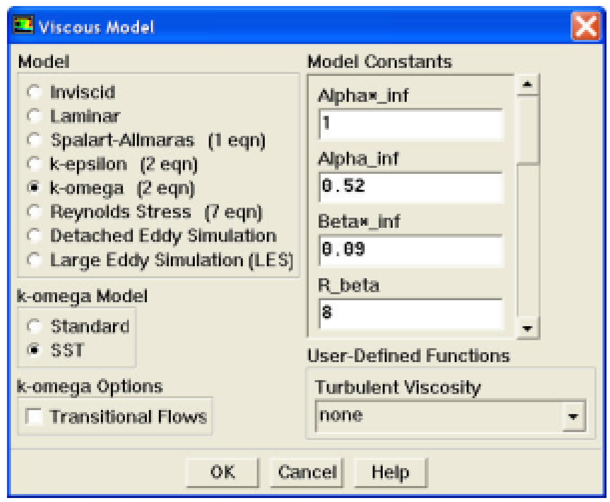
Main Menu > Define > Models > Solver

Choose ***Pressure Based*** under ***Solver***, ***Absolute*** under ***Velocity Formulation***, ***Green-Gauss Node Based*** under ***Gradient Option***, ***Unsteady*** under ***Time*** , ***2nd-Order Implicit*** under ***Unsteady Formulation***, and ***Superficial Velocity*** under ***Porous Formulation***.



Main Menu > Define > Models > Viscous

Choose ***k-omega Model*** and check the ***SST***. Leave all other values alone.



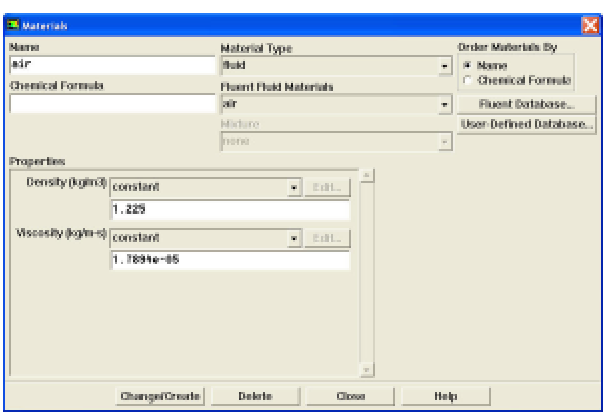
Main Menu > Define > Models > Energy

For incompressible flow, the energy equation is decoupled from the continuity and momentum equations. We need to solve the energy equation only if we are interested in determining the temperature distribution. We will not deal with temperature in this example. So leave the ***Energy Equation*** unselected and click ***Cancel*** to exit the menu.

Define Material Properties

Main Menu > Define > Materials...

The default values are the ones that we are going to use so we do not need to change anything here. These are the values that we specified under Problem Specification.

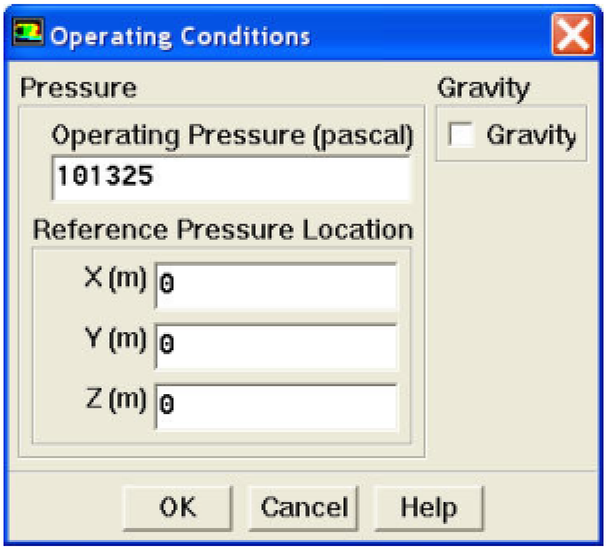


Click ***Change/Create***. Close the window.

Define Operating Conditions

Main Menu > Define > Operating Conditions...

For all flows, FLUENT uses gauge pressure internally. Any time an absolute pressure is needed, it is generated by adding the operating pressure to the gauge pressure. We'll use the default value of 1 atm (101,325 Pa) as the ***Operating Pressure***.

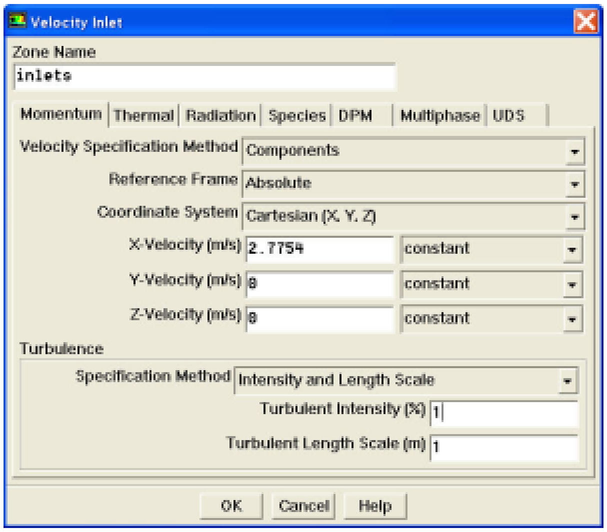


Click ***Cancel*** to leave the default in place.

Define Boundary Conditions

Main Menu > Define > Boundary Conditions...

Select ***Inlets*** and click ***Set***. We should set ***Velocity Specification Method*** to ***Components***, and set ***X-velocity*** to 2.7754. In the ***Turbulence Area***, we should set ***Specification Method*** to ***Intensity*** and ***Length Scale***, and set ***Turbulent Intensity*** to 1.



All other Boundary Conditions have been set during the meshing process so nothing else should be modified here.

Click ***Close*** to close the ***Boundary Conditions*** menu.

Go to Step 5: Solve!
See and rate the complete Learning Module

Go to all FLUENT Learning Modules