Turbulent Pipe Flow - Physics Setup

Author: Rajesh Bhaskaran, Cornell University

Problem Specification
1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Numerical Solution
6. Numerical Results
7. Verification & Validation
Exercises
Comments

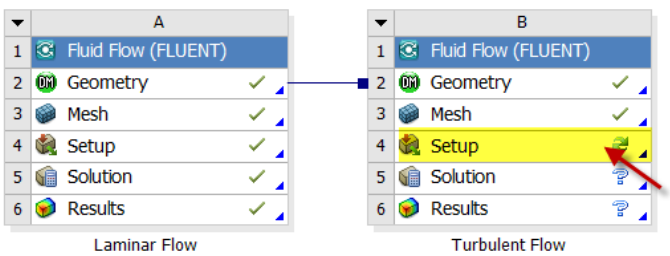
Useful Information

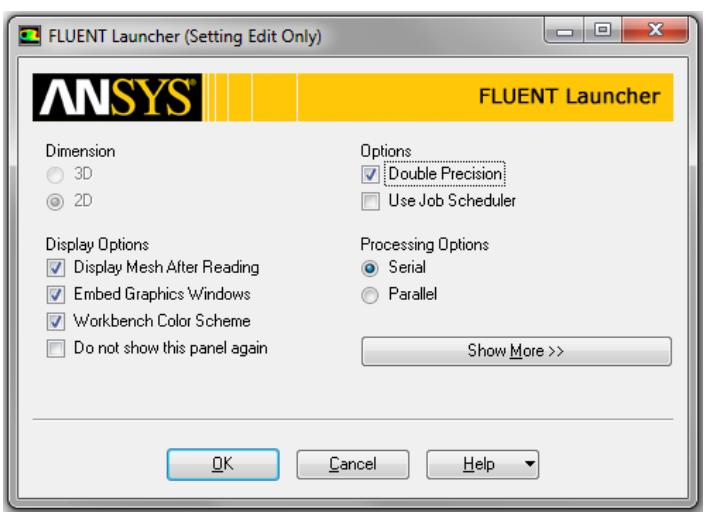
Click here for the FLUENT 6.3.26 version.

Physics Setup

Launch FLUENT

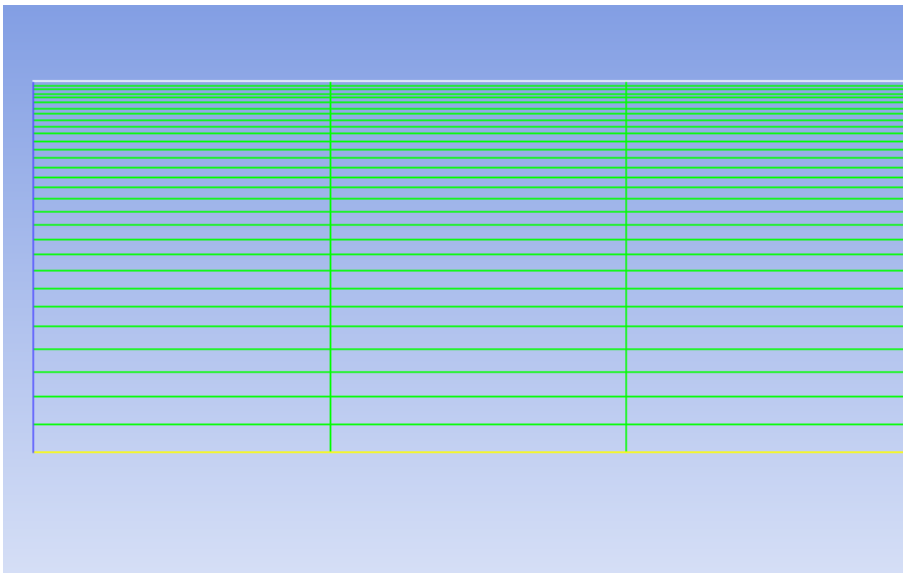
We will be working within ANSYS Workbench. To launch FLUENT, double click on the ***Setup*** cell from the Project view. Make sure the ***Double Precision*** option is selected. This will use 64 bits (rather than 32) per floating point number, decreasing round-off errors.





Once Fluent has opened, select **Problem Setup > General > Display...**

Make sure all 5 items under ***Surfaces*** are selected. Then click ***Display***. Remember that we can zoom in using the middle mouse button. Zoom in and admire the mesh. How many divisions are there in the radial direction?



Recall that you can look at specific components of the mesh by choosing the entities 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. Use this feature and make sure that the boundary labels correspond to the correct geometric entities.

Define Governing Equations

Problem Setup > General > Solver

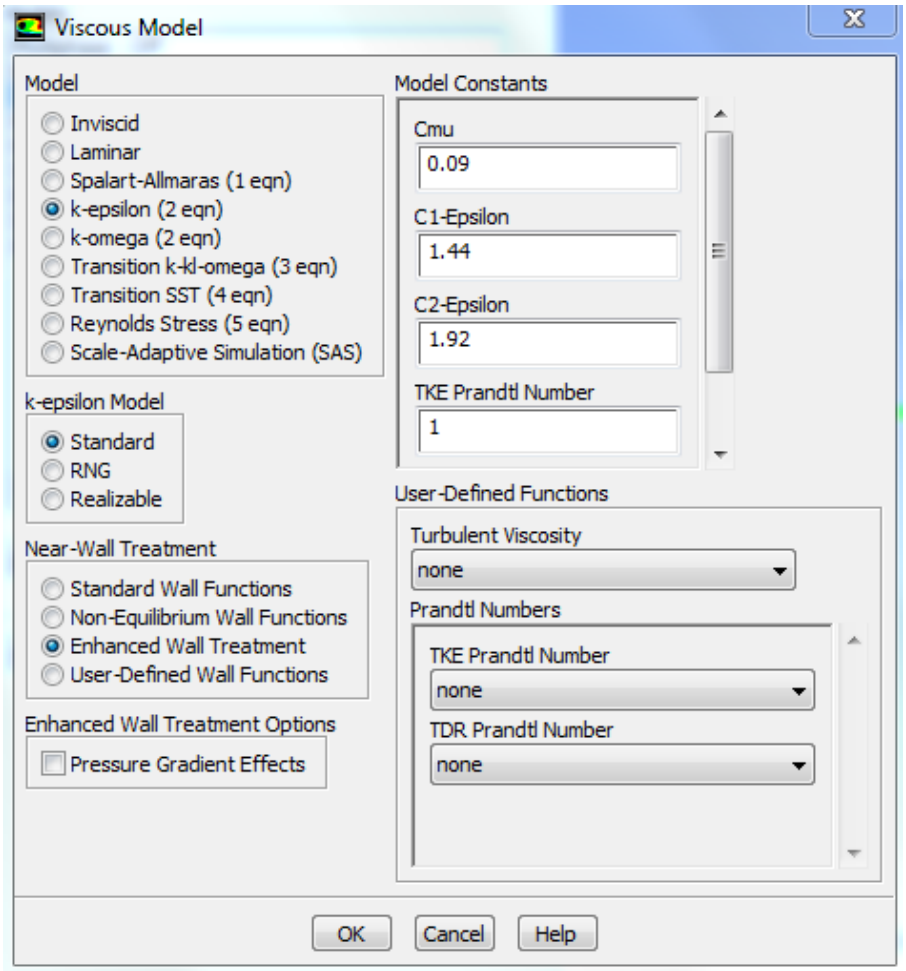
Choose ***Axisymmetric*** under ***2D Space***. As in the laminar pipe flow tutorial, we'll use the defaults of Pressure-Based Type, Steady flow and Absolute Velocity Formulation.

Problem Setup > Models > Energy...

The energy equation can be turned off since this is an incompressible flow and we are not interested in the temperature. Make sure ***Energy - Off*** appears.

Problem Setup > Models > Viscous - Laminar

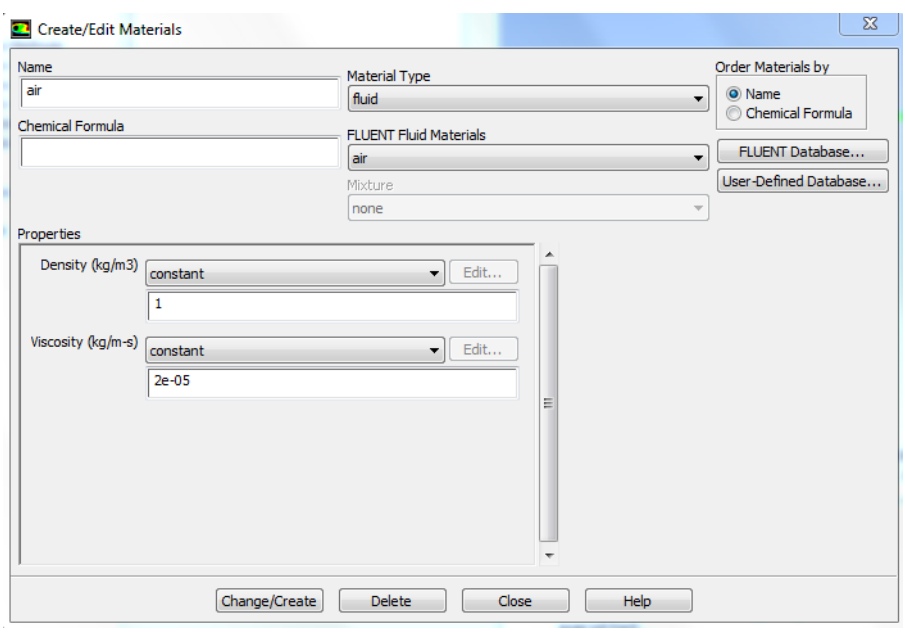
Click ***Edit...*** and choose ***k-epsilon (2eqn)***. Notice that the window expands and additional options are displayed on choosing the *k-epsilon* turbulence model. Under ***Near-Wall Treatment***, pick ***Enhanced Wall Treatment***. This option uses a blended function to go between a two-layer model and standard wall functions. If the mesh near the wall is fine enough, the two-layer model is used. Otherwise, standard wall functions are used. You could alternately use ***Standard Wall Functions***; this will work well when $30 < y+ < 100$. Refer to the turbulence chapter in the FLUENT user manual.



Click ***OK***.

Problem Setup > Materials

Double click on ***air*** and change ***Density*** to `1.0 kg/m^3` and ***Viscosity*** to `2e-5 kg/(m*s)`. These are the values in the Problem Specification and are picked to give us a Reynolds number of 10,000. We'll take both as constant.



Click ***Change/Create*** and close the window.

Define Boundary Conditions

Problem Setup > Boundary conditions > Operating Conditions...

Recall that for all flows, FLUENT uses the 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***.

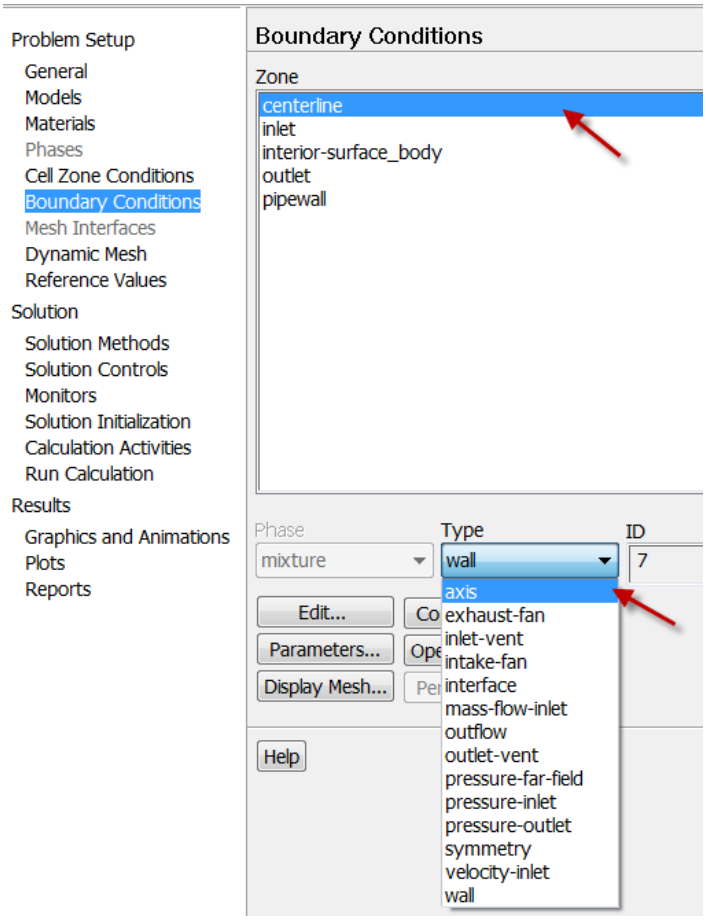
Click ***Cancel*** to leave the default in place.

We'll now setup the boundary conditions at the wall, centerline, inlet and outlet.

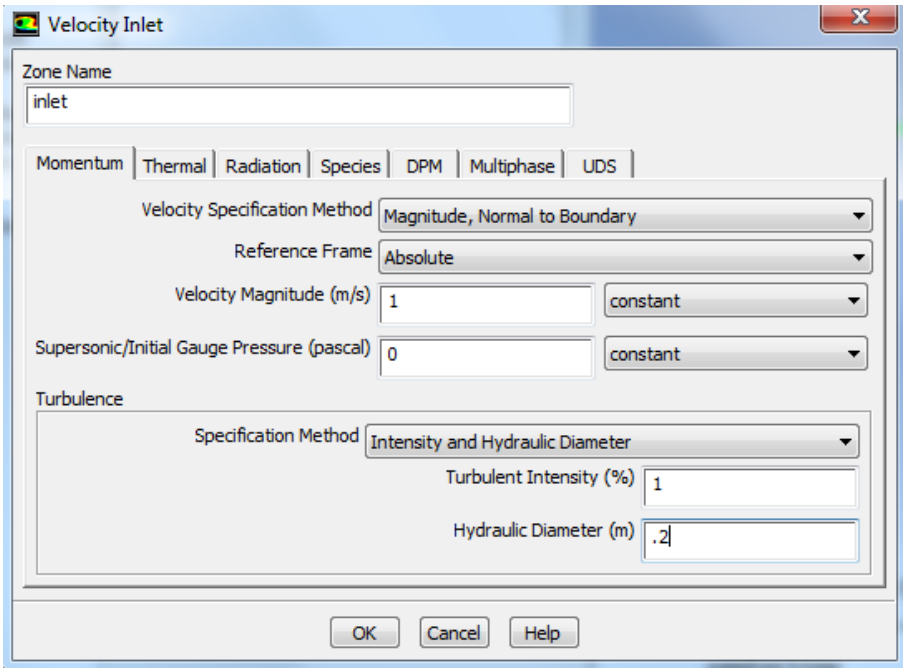
Problem Setup > Boundary conditions

We don't need to set any parameters for the ***pipewall*** zone. FLUENT will automatically detect that this location should be set as a wall based on its name. Verify this by selecting that zone and looking at its type in the drop down menu.

Next, let's look at the centerline. Since we are solving an axisymmetric problem, we will set the centerline as the axis; this will impose symmetry at this boundary. Set ***centerline*** to axis boundary type, using the drop down menu. Click ***Yes*** and ***OK*** to confirm.



Choose inlet and click on ***Edit....***. This boundary is set to ***velocity-inlet*** type by default which is right in our case. Change the ***Velocity Specification Method*** to ***Magnitude, Normal to Boundary.*** Enter `1 m/s`for ***Velocity Magnitude***. This indicates that the fluid is coming in normal to the inlet at the rate of 1 meter per second. Select ***Intensity and Hydraulic Diameter*** next to the ***Turbulence Specification Method***. Then enter `1%` for ***Turbulence Intensity*** and `0.2m` for ***Hydraulic Diameter***. Click ***OK*** to set the boundary conditions for the inlet.



The (absolute) pressure at the outlet is 1 atm. Since the operating pressure is set to 1 atm, the outlet gauge pressure = outlet absolute pressure - operating pressure = 0. Choose ***outlet*** under ***Zone***. The ***Type*** of this boundary is ***pressure-outlet***. Click on ***Edit.*** The default value of the ***Gauge Pressure*** is 0. Click ***Cancel*** to leave the defaults in place.

Note: Backflow in the *Pressure Outlet* menu refers to flow *entering* through an outlet boundary. This is not likely to happen in this case. So we don't have to set the backflow parameters.

This completes the boundary condition specification.

Reference Values

Let's setup the reference values, which will be used later on while viewing non-dimensional results (this setting doesn't affect the numerical solution).

Problem Setup > Reference Values
Select ***Compute from > inlet***.

Go to Step 5: Numerical Solution

Go to all FLUENT Learning Modules