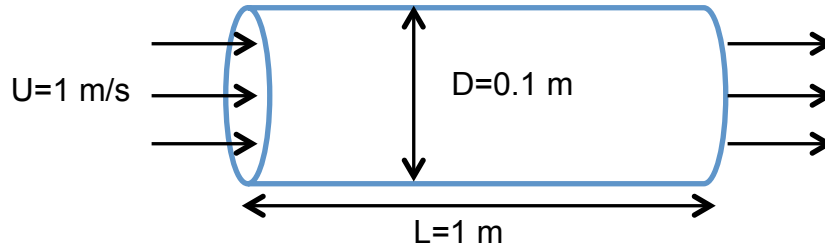


## CSE Training: Fluent

### Laminar flow in a circular tube

#### Problem Description



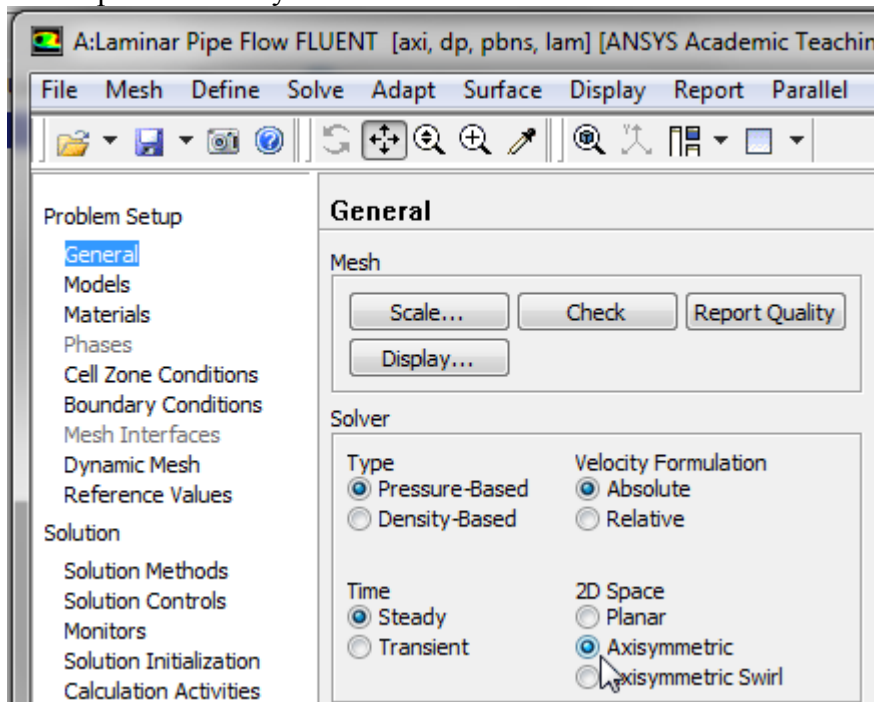
Consider fluid flowing through a circular tube of constant radius. The tube diameter  $D=0.1$  m and length  $L=1$  m. The inlet velocity  $U=1$  m/s. The fluid discharges into the atmosphere at a pressure of 1 atm. Take the density  $\rho=1$  kg/m<sup>3</sup> and the viscosity  $\mu=1\times 10^{-3}$  kg/(ms). The Reynolds number based on the tube diameter is

$$Re = \frac{\rho U D}{\mu} = 100$$

#### Setting up problem in FLUENT

Launch FLUENT and check the “Double Precision” option.

Select “Axisymmetric” under “General > Solver > 2D Space”. Think about why axisymmetric versus planar or axisymmetric swirl.



We are solving an isothermal laminar flow. Neither turbulence model nor energy equation is needed. So leave the “Models” menu be.

Change the properties of air as specified before under the “Materials” menu.

The screenshot shows the 'Create/Edit Materials' dialog box. The 'Name' field contains 'air'. The 'Material Type' dropdown is set to 'fluid'. The 'Chemical Formula' field is empty. The 'Fluent Fluid Materials' dropdown is set to 'air'. The 'Mixture' dropdown is set to 'none'. The 'Order Materials by' section has 'Name' selected. The 'Properties' section shows 'Density (kg/m³)' set to 'constant' with a value of 1, and 'Viscosity (kg/m-s)' set to 'constant' with a value of 0.001. There are 'Edit...' buttons for both properties. At the bottom are buttons for 'Change/Create', 'Delete', 'Close', and 'Help'.

Skip the “Cell Zone Conditions” as we do not add any new materials. No update here is needed.

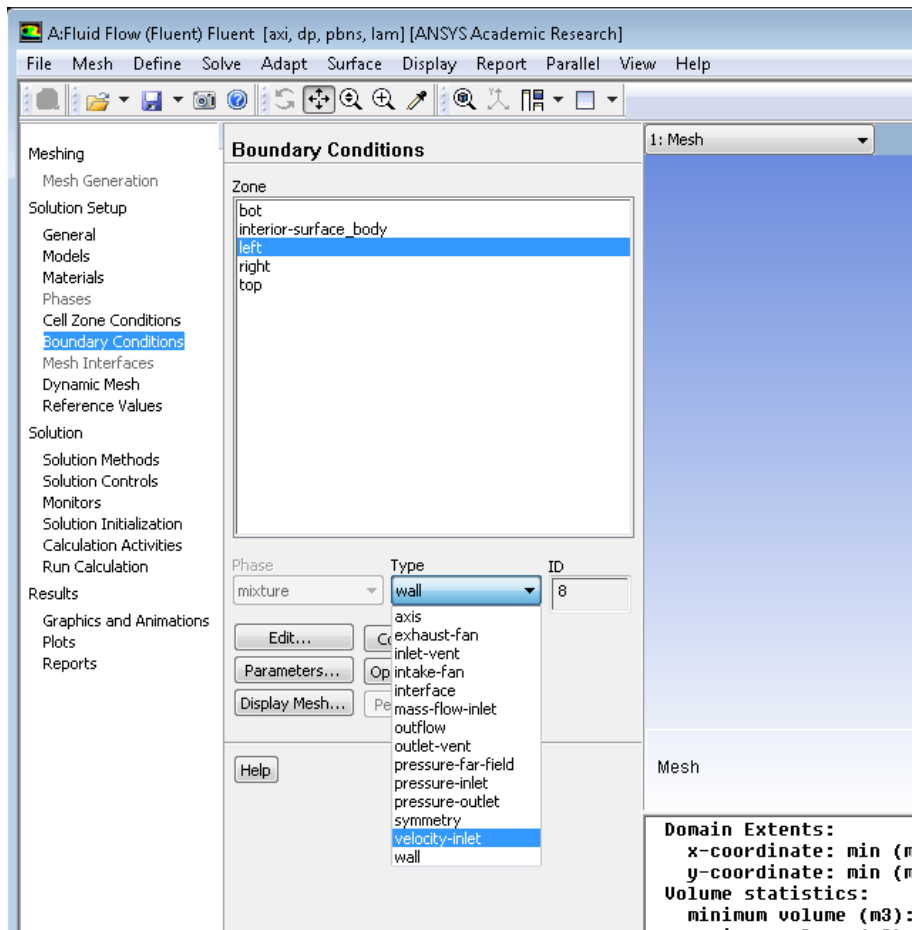
In “Boundary Conditions”, you will see four boundaries labeled as left, right, top, and bottom, and an interior zone. The left boundary will serve as the inlet, right as the outlet, top as the wall, and bottom as the symmetric axis.

You can choose the type of a boundary by the “Type” pull down list. For example, select “velocity-inlet” for the left boundary. Then enter 1 m/s for the velocity magnitude.

Select “pressure-outlet” for the right boundary. Keep the gauge pressure as 0 pascal. Why not 101325 pa?

Select “axis” for the top boundary. Why not symmetry? What is the difference?

The bottom boundary is a wall by default. So leave it be.



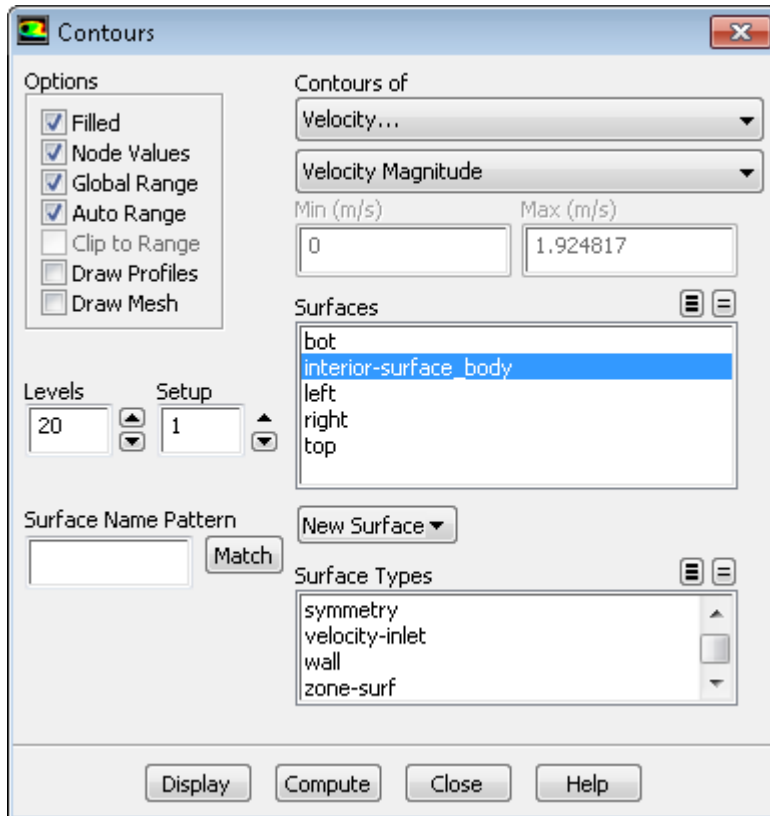
Go to “Solution Initialization”. Choose “Standard Initialization”, select “Compute from” “left”, and click on “Initialize”. Why computing from left? What if you initialize from other regions?

After initialization, go to “Run calculation” and carry out 100 iterations. It should converge very quickly.

## Post-processing

Various visualization tools are available in the “Results” menu. You can draw contours, vectors, and streamlines from the “Graphics and Animations” menu. For example, make a contour of velocity on the entire surface. Double click on “Contours”. Change the “Contours of” pull down list to “Velocity”. Check the “Filled” box. Click “Display”. Does the contour plot make sense to you?

Try making a vector plot from “Vectors” and a streamline plot from “Pathlines”.



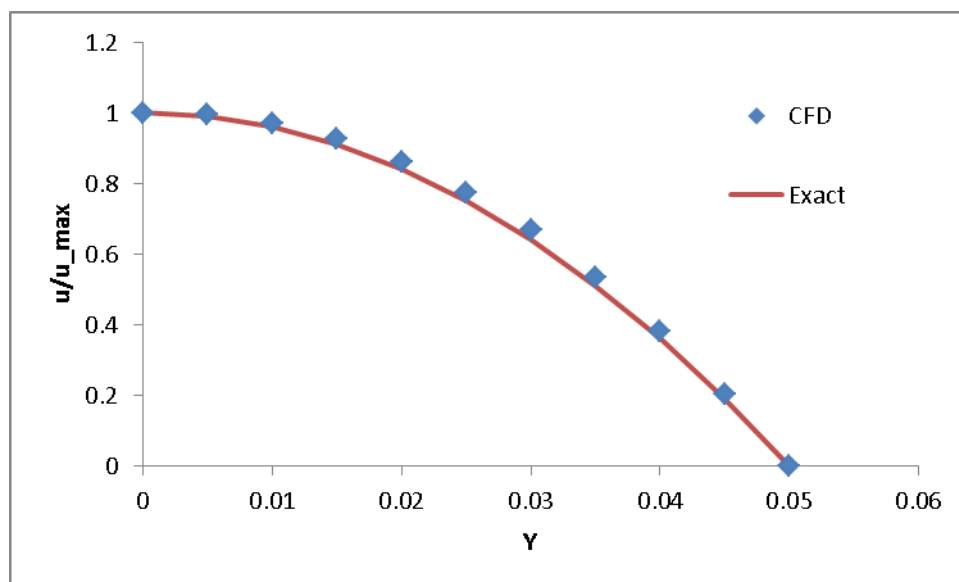
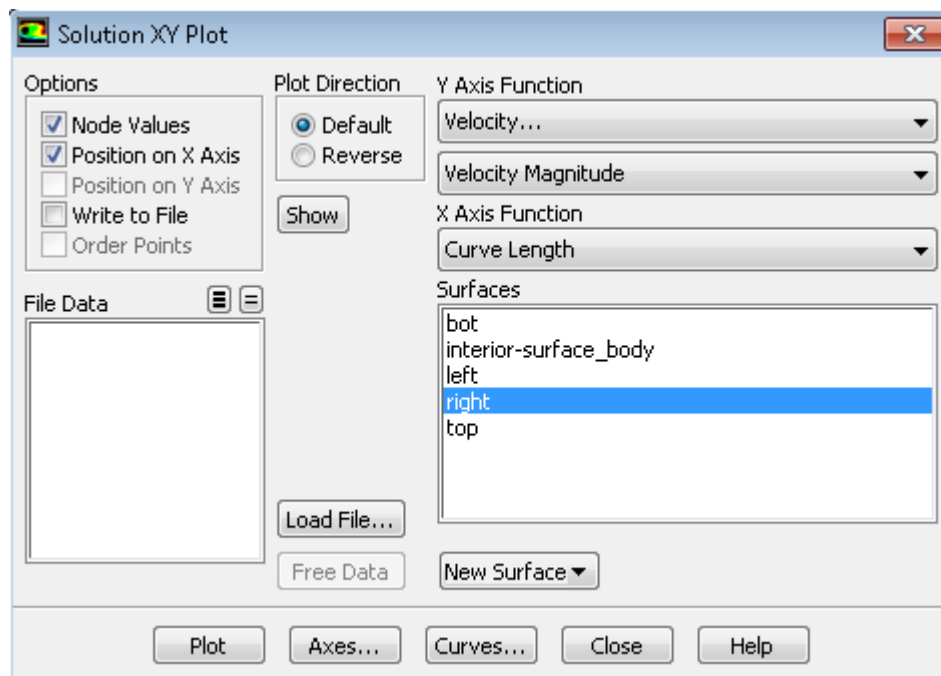
These plots, however, give us limited information regarding to the question “how do I know this is a good simulation?”. So let’s look at the actual computed values.

Go to “Plots” in the navigation pane, and double click on “XY Plot”. In the window that appears, change “Y Axis Function” to velocity, and change “X Axis Function” to Curve Length. First plot the velocity along the central axis. Select “bottom” from the Surfaces list, and click on “Plot”. Has the flow become fully developed? Next, plot the velocity profile along the outlet. Is this a good simulation?

You can also export the data to a txt file so that you can compare it with the exact solution:

$$u(r) = u_{max} \left(1 - \frac{r^2}{R^2}\right)$$

Check the “Write to File” box in the upper left corner. Click on “Write” and name the file of output. Copy the output data to Excel and make a plot of  $u/u_{max}$  versus  $Y$ .



### Revise the setup to simulate a 2D lid-driven cavity flow

Since we take advantage of the axisymmetric nature of circular pipe, it is easy to transform the pipe flow problem to a lid-driven cavity one.

Go back to “General”, and change back to a “Planar” “2D Space”.

Another thing you need to change is boundary conditions. In a lid-driven cavity, all four boundaries are walls. The top wall is sliding at a constant velocity of 1 m/s in X direction.

Edit the top boundary. Select “Moving Wall” in the “Wall Motion” option, and enter “1” for the speed. Accept other settings.

Now re-initialize and run the simulation again. Once you have the results, repeat the post-processing steps. Is it a good simulation?

**Wall**

Zone Name  
top

Adjacent Cell Zone  
surface\_body

Momentum | Thermal | Radiation | Species | DPM | Multiphase | UDS | Wall Film

**Wall Motion**

☐ Stationary Wall  
☒ Moving Wall

**Motion**

☒ Relative to Adjacent Cell Zone  
☐ Absolute

Speed (m/s)  
1

**Direction**

X 1  
Y 0

**Shear Condition**

☒ No Slip  
☐ Specified Shear  
☐ Specularity Coefficient  
☐ Marangoni Stress

**Wall Roughness**

Roughness Height (m) 0 constant  
Roughness Constant 0.5 constant

OK Cancel Help