FLUENT - Turbulent Pipe Flow - Step 6

Useful Information

Click here for the FLUENT 12 version.

Step 6: Results

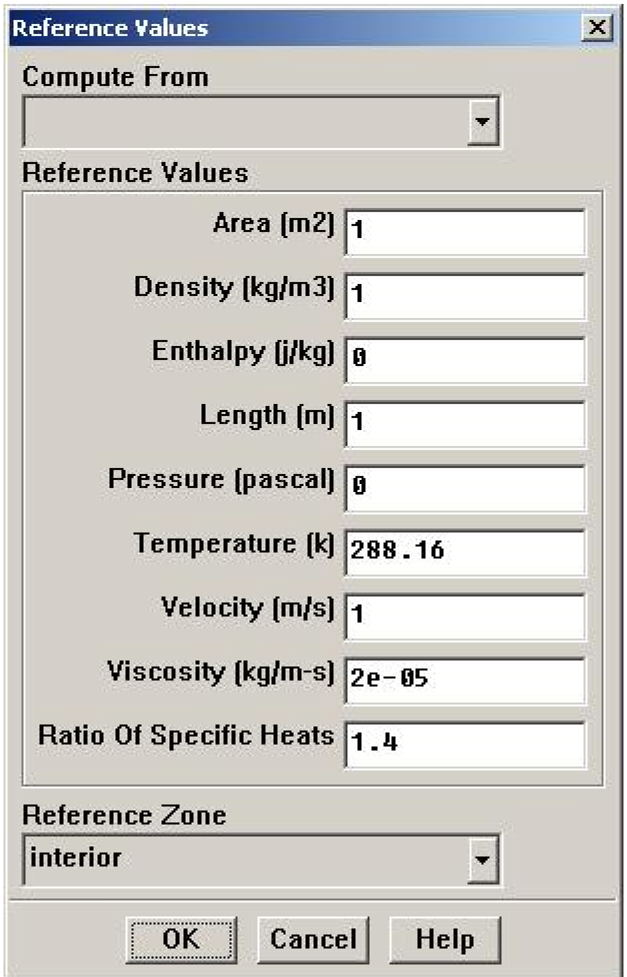
y+

Turbulent flows are significantly affected by the presence of walls. The *k-epsilon* turbulence model is primarily valid away from walls and special treatment is required to make it valid near walls. The near-wall model is sensitive to the grid resolution which is assessed in the wall unit *y+* (defined in section 10.9.1 of the FLUENT user manual). We'll gloss over the details for now and use the following rule of thumb: select the near-wall resolution such that *y+ > 30* or *< 5* for the wall-adjacent cell. Look at section 10.9, *Grid Considerations for Turbulent Flow Simulations*, for details.

First, we need to set the reference values needed to calculate *y+*.

Main Menu > Report > Reference Values...

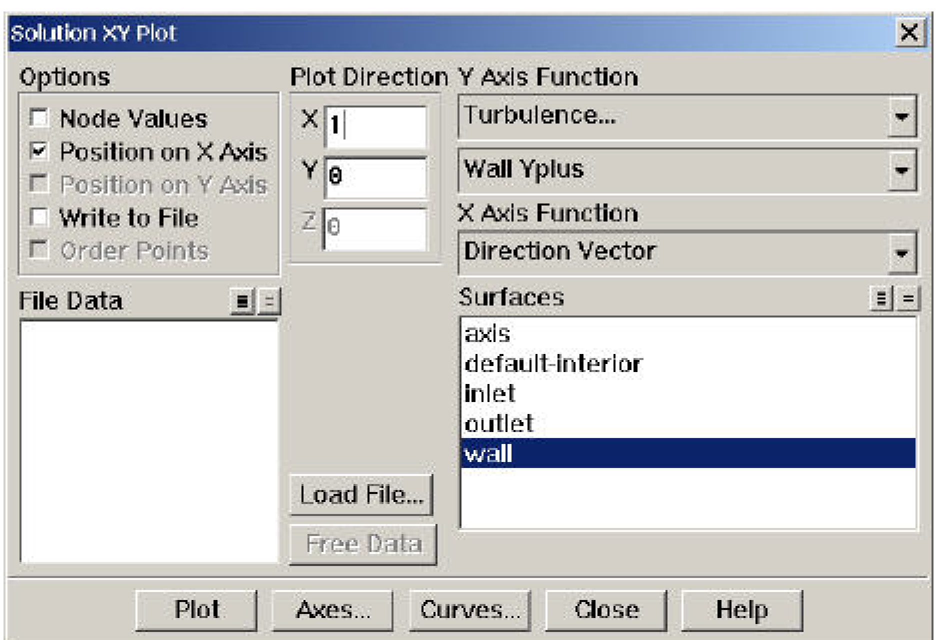
Select ***inlet*** under ***Compute From*** to tell FLUENT to use values at the pipe inlet for the reference values. Check that the reference value for density is $1\ kg/m^3$, velocity is *1 m/s,* and coefficient of viscosity is *2e-5 kg/m-s* as given in the Problem Specification. These reference values will be used to non-dimensionalize the distance of the cell center from the wall to obtain the corresponding *y+* values. Click ***OK***.



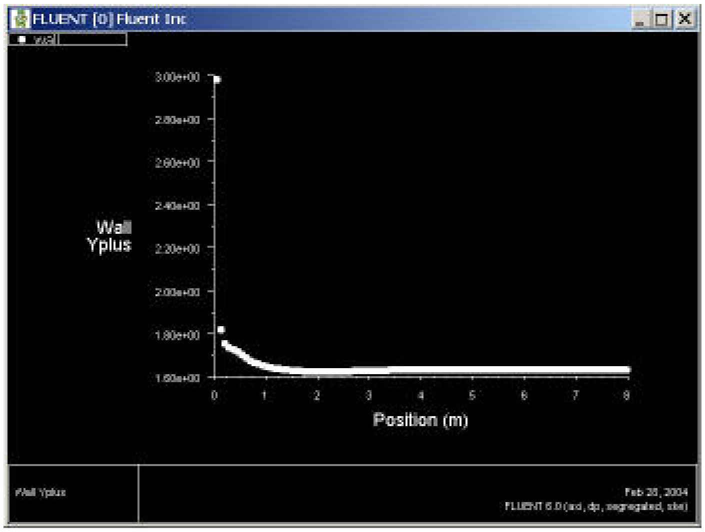
Let's plot *y+* values for wall-adjacent cells to check how it compares with the recommendation mentioned above.

Main Menu > Plot > XY Plot...

Make sure that ***Position on X Axis*** is set under ***Options***, that 1 is the value next to ***X***, and 0 is the value next to ***Y*** and ***Z*** under ***Plot Direction***. Recall that this tells FLUENT to plot the x-coordinate value on the abscissa of the graph. Pick ***Turbulence...*** under ***Y Axis Function*** and select ***Wall Yplus*** from the drop down list under that. Since we want the *y+* value for cells adjacent to the wall of the pipe, choose wall under ***Surfaces***.



Click ***Plot***.



Higher Resolution Image

As we can see, the wall *y*+ value is between 1.6 and 1.9 (ignoring the anamolous at the inlet). Since this is less than 5, the near-wall grid resolution is acceptable.

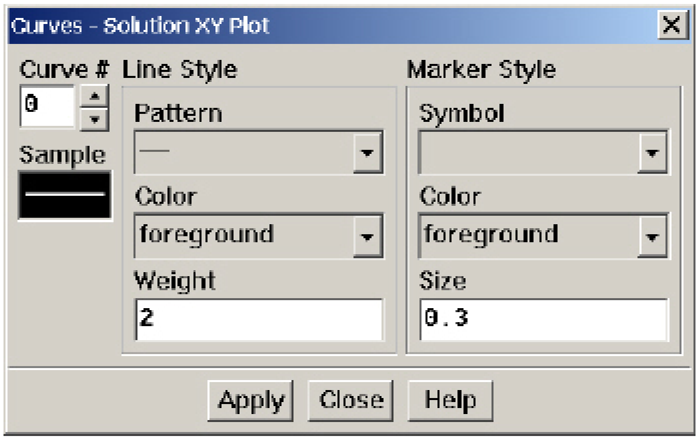
Save Plot

In the *Solution XY Plot Window*, check the ***Write to File*** box under Options. The ***Plot*** button should have changed to the ***Write...*** button. Click on ***Write....*** Enter `yplus.xy` as the filename and click ***OK***. Check that this file has been created in your FLUENT working directory.

Centerline Velocity

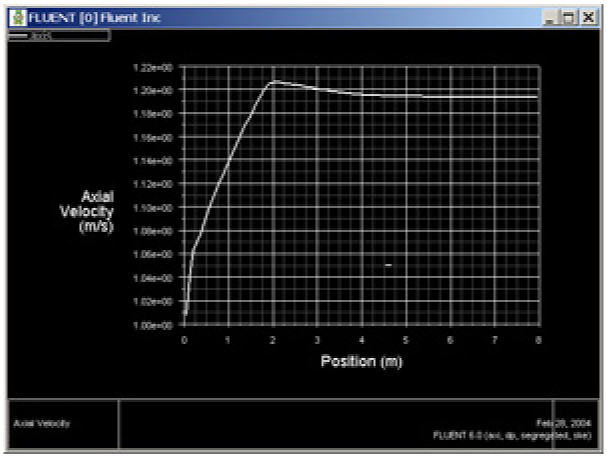
Under ***Y Axis Function***, pick ***Velocity***... and then in the box under that, pick ***Axial Velocity***. Finally, select ***centerline*** under ***Surfaces*** since we are plotting the axial velocity along the centerline. De-select ***wall*** under ***Surfaces***.

Click on ***Curves***... in the *Solution XY Plot* window. Select the solid line option under ***Pattern*** as shown below. Change ***Weight*** to 2. Select the blank option under ***Symbol***. Click ***Apply*** and ***Close***.



Turn on grid lines: In the *Solution XY Plot* window, click on ***Axes....*** Turn on the grid by checking the boxes ***Major Rules*** and ***Minor Rules*** under ***Options***. Leave Auto Range checked. Click ***Apply***. Select ***Y*** under ***Axis*** and repeat. Click ***Apply*** and ***Close***.

Uncheck ***Write to File***. Click ***Plot***.



Higher Resolution Image

We can see that the fully developed region starts around *x*=5m with the centerline velocity becoming constant at a value of 1.195 m/s. This is quite a bit lower than the value of 2 m/s for the laminar case. Can you explain the difference based on the physical characteristics of laminar and turbulent flows?

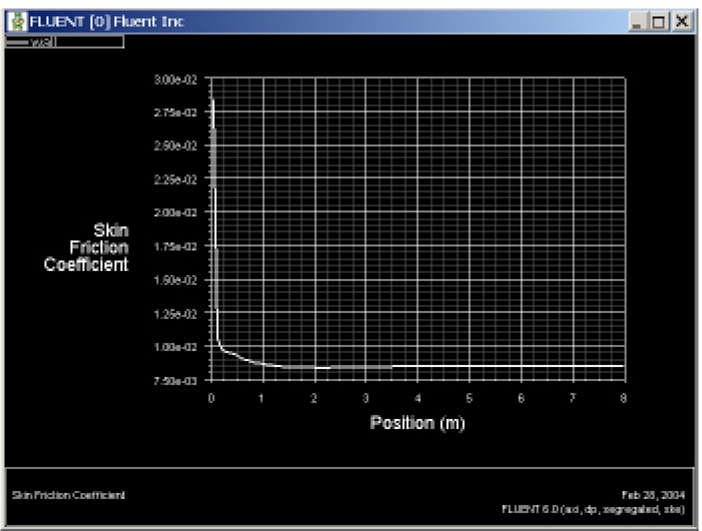
Save the data for this plot as vel.xy.

Coefficient of Skin Friction

The definition of the skin friction coefficient was discussed in the laminar pipe flow tutorial. The required reference values of density and velocity have already been set when plotting *y+*.

Go back to the *Solution XY Plot Window*. Under the ***Y Axis Function***, pick ***Wall Fluxes...***, and then ***Skin Friction Coefficient*** in the box under that. Under ***Surfaces***, we are plotting the friction coefficient along the ***wall***. Uncheck ***centerline*** surface.

Uncheck ***Write to File***. Click ***Plot***.



Higher Resolution Image

We can see that the fully-developed value is *0.0085*. Compare this with what you'd expect from the Moody chart.

Save the data for this plot as cf.xy.

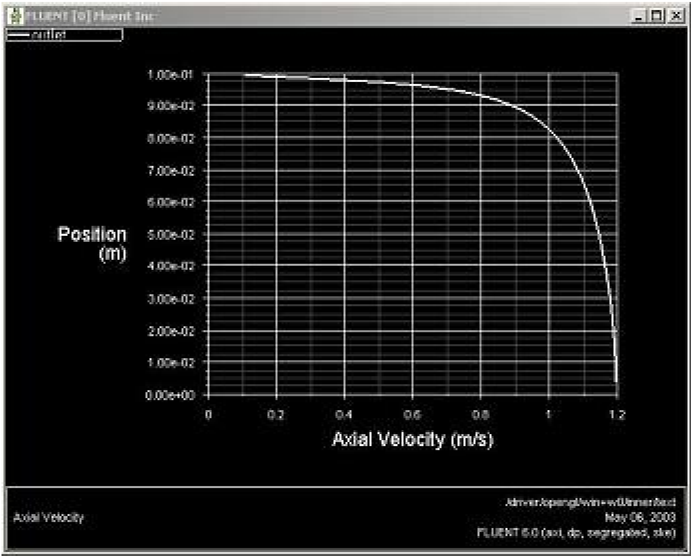
Velocity Profile

We'll plot the axial velocity at the outlet as a function of the distance from the center of the pipe.

Change the plot settings so that the radial distance from the axis is plotted as the ordinate: In the *Solution XY Plot* window, uncheck ***Position on X*** Axis under ***Options*** and choose ***Position on Y Axis*** instead. Under ***Plot Direction***, change ***X*** to `0` and ***Y*** to `1`. For the ***X Axis Function*** i.e. the abscissa, pick ***Velocity...*** and ***Axial Velocity*** under that.

Since we want to plot this at the outlet boundary, pick only ***outlet*** under ***Surfaces***.

Uncheck ***Write to File***. Click ***Plot***.



Higher Resolution Image

The axial velocity is maximum at the centerline and zero at the wall to satisfy the no-slip boundary condition for viscous flow. Compare qualitatively the near-wall velocity gradient normal to the wall with the laminar case. Which is larger? From this, what can you say about the relative strengths of near-wall mixing in the laminar and turbulent cases?

Save this plot as `profile.xy`.

Go to Step 7: Verification & Validation

See and rate the complete Learning Module

Go to all FLUENT Learning Modules