

FLUENT - Laminar Pipe Flow Step 6

Problem Specification
1. Pre-Analysis & Start-up
2. Mesh
3. Geometry
4. Setup (Physics)
5. Solution
6. Analysis & Results
7. Verification & Validation
Problem 1
Problem 2



Useful Information

[Click here](#) for the FLUENT 12

Step 6: Analysis & Results

Centerline Velocity

We'll plot the variation of the axial velocity along the centerline.

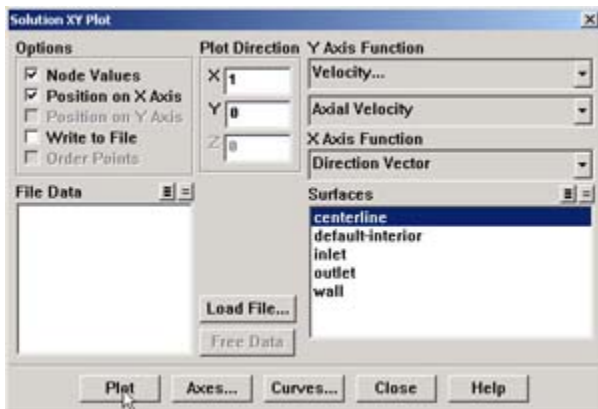
Main Menu > Plot > XY Plot...

Make sure that **Position on X Axis** is set under **Options**, and **X** is set to 1 and **Y** to 0 under **Plot Direction**. This tells FLUENT to plot the x-coordinate value on the abscissa of the graph.

Under **Y Axis Function**, pick **Velocity...** and then in the box under that, pick **Axial Velocity**.

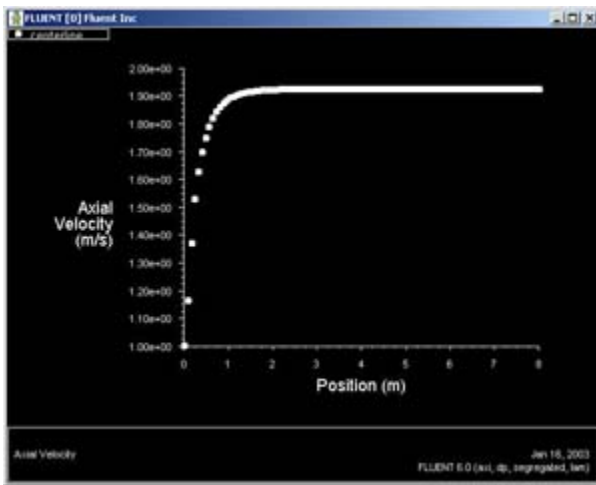
Please note that **X Axis Function** and **Y Axis Function** describe the x and y axes of the *graph*, which should not be confused with the x and y directions of the *pipe*.

Finally, select **centerline** under **Surfaces** since we are plotting the axial velocity along the centerline. This finishes setting up the plotting parameters.



Click **Plot**.

This brings up a plot of the axial velocity as a function of the distance along the centerline of the pipe.



[Higher Resolution Image](#)

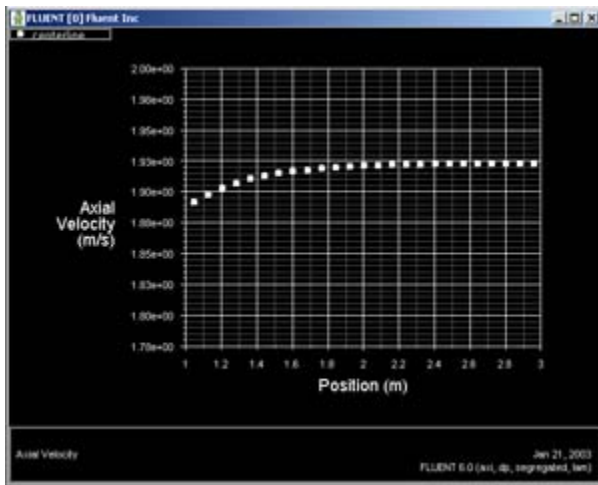
In the graph that comes up, we can see that the velocity reaches a constant value beyond a certain distance from the inlet. This is the fully-developed flow region.

Change the axes extents: In the *Solution XY Plot* window, click on **Axes....** Under **Options**, deselect **Auto Range**. The boxes under **Range** should now be activated. Select **X** under **Axis**. Enter 1 for **Minimum** and 3 for **Maximum** under **Range**.

We'll turn on the grid lines to help estimate where the flow becomes fully developed. Check the boxes next to **Major Rules** and **Minor Rules** under **Options**. Click **Apply**.

Now, pick **Y** under **Axis** and once again deselect **Auto Range** under **Options**, then enter 1.8 for **Minimum** and 2.0 for **Maximum** under **Range**. Also select **Major Rules** and **Minor Rules** to turn on the grid lines in the direction. We have now finished specifying the range for each axes, so click **Apply** and then **Close**.

Go back to the *Solution XY Plot* menu and click **Plot** to replot the graph with the new axes extents. We can see that the fully-developed region starts at around $x=3\text{m}$ and the centerline velocity in this region is 1.93 m/s .



[Higher Resolution Image](#)

Saving the Plot

Save the data from this plot:

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

Now, save a picture of the plot:

Leave the *Solution XY Plot Window* and the *Graphics Window* open and in the main FLUENT window click on:

File > Hardcopy ...

Under **Format**, choose one of the following three options:

EPS - if you have a postscript viewer, this is the best choice. **EPS** allows you to save the file in vector mode, which will offer the best viewable image quality. After selecting **EPS**, choose **Vector** from under **File Type**.

TIFF - this will offer a high resolution image of your graph. However, the image file generated will be rather large, so this is not recommended if you do not have a lot of room on your storage device.

JPG - this is small in size and viewable from all browsers. However, the quality of the image is not particularly good.

After selecting your desired image format and associated options, click on **Save...**

Enter `vel.eps`, `vel.tif`, or `vel.jpg` depending on your format choice and click **OK**.

Verify that the image file has been created in your working directory. You can now copy this file onto a disk or print it out for your records.

Coefficient of Skin Friction

FLUENT provides a large amount of useful information in the online help that comes with the software. Let's probe the online help for information on calculating the coefficient of skin friction.

Main Menu > Help > User's Guide Index...

Click on S in the links on top and scroll down to **skin friction coefficient**. Click on the second link (normally, you would have to go through each of the links until you find what you are looking for). We can see an excerpt on the skin coefficient as well as the equation for calculating it.

Click on the link for **Reference Values** panel, which tells us how to set the reference values used in calculating the skin coefficient.

Skin Friction Coefficient

(in the **Wall Fluxes...** category) is a nondimensional parameter defined as the ratio of the wall shear stress and the reference dynamic pressure

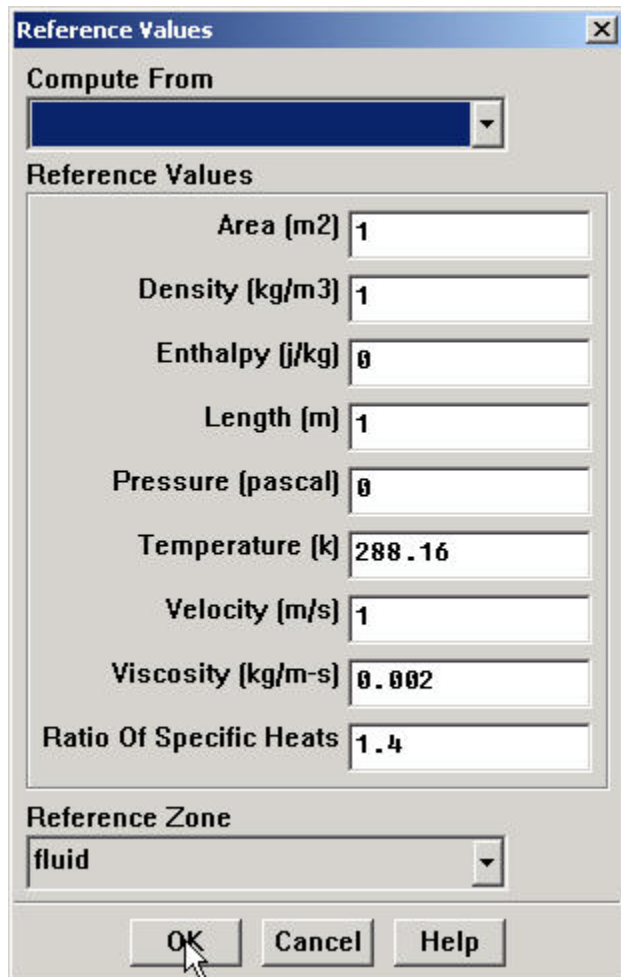
$$C_f \equiv \frac{\tau_w}{\frac{1}{2} \rho_{\text{ref}} v_{\text{ref}}^2} \quad (27.4.26)$$

where τ_w is the wall shear stress, and ρ_{ref} and v_{ref} are the reference density and velocity defined in the [Reference Values panel](#).

Set the reference values:

Main Menu > Report > Reference Values...

Select **inlet** under **Compute From** to tell FLUENT to calculate the reference values from the values at inlet. Check that density is 1 kg/m^3 and velocity is 1 m/s . (Alternately, you could have just typed in the appropriate values). Click **OK**.



The image shows the 'Reference Values' dialog box in FLUENT. The 'Compute From' dropdown is set to 'inlet'. The 'Reference Values' section contains the following fields:

Parameter	Value
Area (m2)	1
Density (kg/m3)	1
Enthalpy (j/kg)	0
Length (m)	1
Pressure (pascal)	0
Temperature (k)	288.16
Velocity (m/s)	1
Viscosity (kg/m-s)	0.002
Ratio Of Specific Heats	1.4

The 'Reference Zone' dropdown is set to 'fluid'. At the bottom are 'OK', 'Cancel', and 'Help' buttons. A mouse cursor is pointing at the 'OK' button.

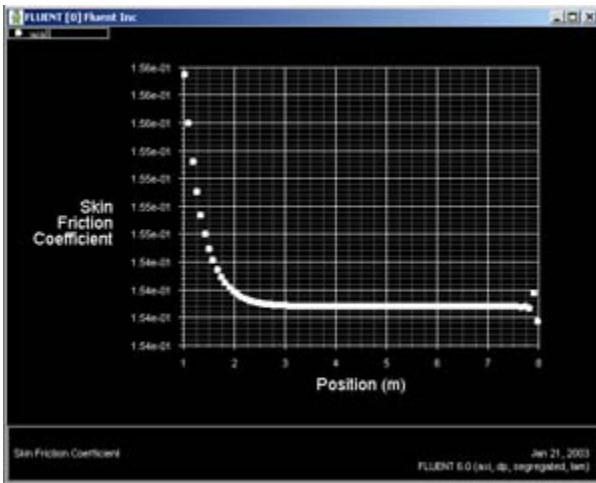
Go back to the **Solution XY Plot** menu. Uncheck **Write to File** under **Options** since we want to plot to the window right now. We can leave the other **Option** and **Plot Direction** as is since we are still plotting against the x distance along the pipe.

Under the **Y Axis Function**, pick **Wall Fluxes...**, and then **Skin Friction Coefficient** in the box under that.

Under **Surfaces**, select **wall** and unselect **centerline** by clicking on them.

Reset axes ranges: Go to **Axes...** and re-select **Auto-Range** for the **Y** axis. Click **Apply**. Set the range of the **X** axis from 1 to 8 by selecting **X** under **Axis**, entering 1 under **Minimum**, and 8 under **Maximum** in the box (remember to de-select **RangeAuto-Range** first if it is checked).

Click **Apply**, **Close**, and then **Plot** in the **Solution XY Plot Window**.



[Higher Resolution Image](#)

We can see that the fully developed region is reached at around $x=3.0\text{m}$ and the skin friction coefficient in this region is around 1.54. Compare the numerical value of 1.54 with the theoretical, fully-developed value of 0.16.

Save the data from this plot: Pick **Write to File** under **Options** and click **Write....** Enter `cf.xy` for **XY File** and click **OK**.

Velocity Profile

We'll next plot the velocity at the outlet as a function of the distance from the center of the pipe. To do this, we have to set the y axis of the graph to be the y axis of the pipe (the radial direction).

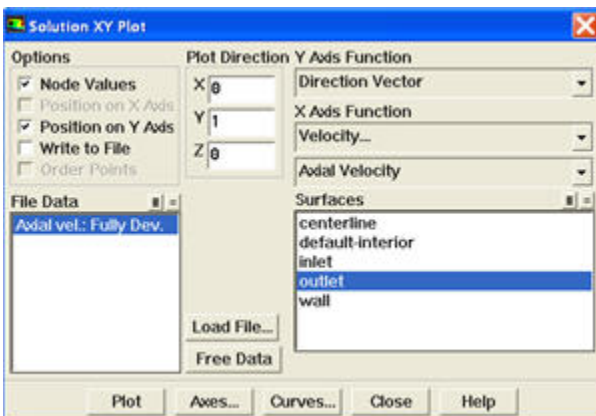
To plot the position variable on the y axis of the graph, uncheck **Position on X Axis** under **Options** and choose **Position on Y Axis** instead. To make the position variable the radial distance from the centerline, under **Plot Direction**, change **X** to 0 and **Y** to 1. To plot the axial velocity on the x axis of the graph, for **X Axis Function**, pick **Velocity...** and **Axial Velocity** under that.

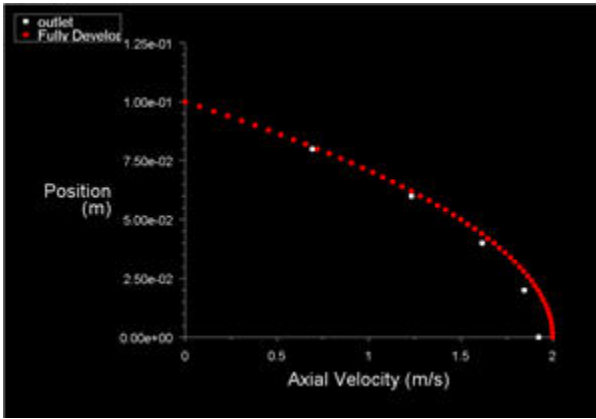
Since we want to plot this at the outlet boundary, pick **outlet** under **Surfaces**.

Change both the x and y axes to **Auto-Range**. (Don't forget to click apply before selecting a different axis)

To compare the velocity profile with theoretical parabolic profile, click **Load File...** and select [profile_fdev.xy](#)

Uncheck **Write to File** under **Options** so that we can see the graph. Click **Plot**.





[Higher Resolution Image](#)

Does this compare well with the theoretical parabolic profile?

Save the data from this plot: Pick **Write to File** under **Options** and click **Write....** Enter `profile.xy` for **XY File** and click **OK**.

To see how the velocity profile changes in the developing region, let us add the profiles at $x=0.6\text{m}$ ($x/D=3$) and $x=0.12\text{m}$ ($x/D=6$) to the above plot. First, create a line at $x=0.6\text{m}$ using the **Line/Rake** tool:

Main Menu > Surface > Line/Rake

We'll create a straight line from $(x_0, y_0)=(0.6, 0)$ to $(x_1, y_1)=(0.6, 0.1)$. Select **Line Tool** under **Options**. Enter $x_0=0.6$, $y_0=0$, $x_1=0.6$, $y_1=0.1$. Enter `line1` under **New Surface Name**. Click **Create**.

To see the line just created, select

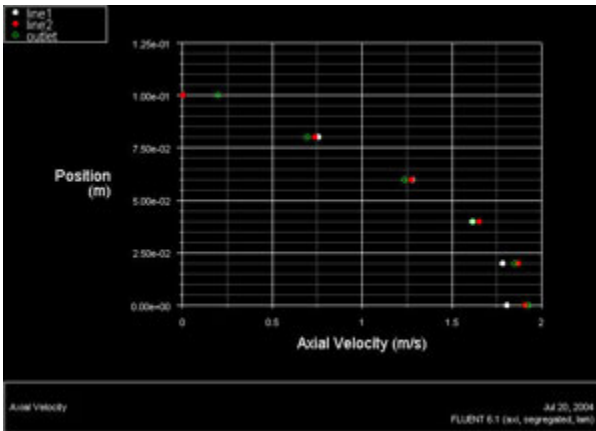
Main Menu > Display > Grid...

Note that **line1** appears in the list of surfaces. Select all surfaces except **default-interior**. Click **Display**. This displays all surfaces but not the mesh cells. Zoom into the region near the inlet to see the line created at $x=0.6\text{m}$. (Click [here](#) to review the zoom functionality discussion in step 4.) **line1** is the white vertical line to the right in the figure below.



Similarly, create a vertical line called *line2* at $x=1.2$; $(x_0,y_0)=(1.2,0)$ to $(x_1,y_1)=(1.2,0.1)$ in this case. Display it in the graphics window to check that it has been created correctly.

Now we can plot the velocity profiles at $x=0.6\text{m}$ ($x/D=3$) and $x=0.12\text{m}$ ($x/D=6$) along with the outlet profile. In the *Solution XY plot* menu, use the same settings as [above](#). Under *Surfaces*, in addition to *outlet*, select *line1* and *line2*. Make sure *Node Values* is selected under Options. Click *Plot*. Your symbols might be different from the ones below. You can change the symbols and line styles under the *Curves...* button. Click on *Help* in the *Curves* menu if you have problems figuring out how to change these settings.



[Higher Resolution Image](#)

The profile three diameters downstream is fairly close to the fully-developed profile at the outlet. If you redo this plot using the fine grid results in the next step, you'll see that this is not actually the case. The coarse grid used here doesn't capture the boundary layer development properly and underpredicts the development length.

In FLUENT, you can choose to display the computed cell-center values or values that have been interpolated to the nodes. By default, the Node Values option is turned on, and the interpolated values are displayed. Node-averaged data curves may be somewhat smoother than curves for cell values.

Velocity Vectors

One can plot vectors in the entire domain, or on selected surfaces. Let us plot the velocity vectors for the entire domain to see how the flow develops downstream of the inlet.

Main Menu > Display > Vectors... > Display

Zoom into the region near the inlet. (Click [here](#) to review the zoom functionality discussion in step 4.) The length and color of the arrows represent the velocity magnitude. The vector display is more intelligible if one makes the arrows shorter as follows: Change *Scale* to 0.4 in the *Vectors* menu and click *Display*.

You can reflect the plot about the axis to get an expanded sectional view:

Main Menu > Display > Views...

Under *Mirror Planes*, only the *axis* surface is listed since that is the only symmetry boundary in the present case. Select *axis* and click *Apply*. Close the *Views* window.



[Higher Resolution Image](#)

The velocity vectors provide a picture of how the flow develops downstream of the inlet. As the boundary layer grows, the flow near the wall is retarded by viscous friction. Note the sloping arrows in the near wall region close to the inlet. This indicates that the slowing of the flow in the near-wall region results in an injection of fluid into the region away from the wall to satisfy mass conservation. Thus, the velocity outside the boundary layer increases.

By default, one vector is drawn at the center of each cell. This can be seen by turning on the grid in the vector plot: Select **Draw Grid** in the *Vectors* menu and then click **Display** in the *Grid Display* as well as the *Vectors* menus. Velocity vectors are the default, but you can also plot other vector quantities. See section 27.1.3 of the user manual for more details about the vector plot functionality.

Go to [Step 7: Refine Mesh](#)

[See and rate the complete Learning Module](#)

[Go to all FLUENT Learning Modules](#)