

Laminar Pipe Flow - Results (FLUENT Post-processor)

Author: Rajesh Bhaskaran, John Singleton, Cornell University

Problem Specification

1. Pre-Analysis & Start-Up

2. Geometry

3. Mesh

4. Setup (Physics)

5. Solution

6. Results

7. Verification & Validation

Exercises



Useful Information

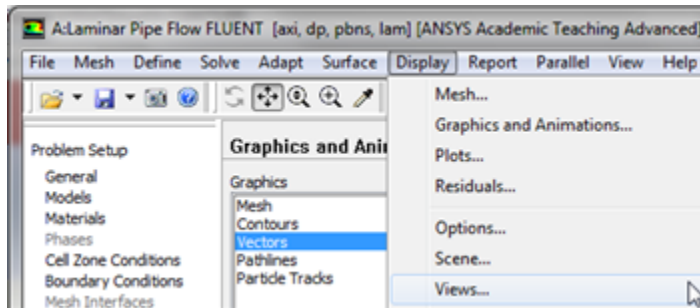
[Click here](#) for the FLUENT 6.3.26 version.

Step 6: Results

Velocity Vectors

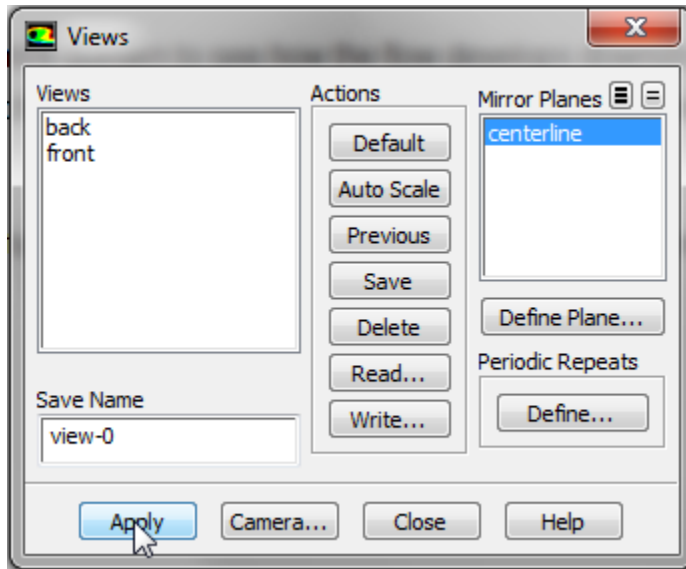
One can plot vectors in the entire domain, or on selected surfaces. Let us plot the velocity vectors for the entire domain to see how the flow develops downstream of the inlet. First, click on **Graphics & Animations**. Next, double click on **Vectors** which is located under **Graphics**. Then, click on **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**.

The laminar pipe flow was modeled asymmetrically; however, the plot can be reflected about the axial axis to get an expanded sectional view. In order to carry this out (**Click** *Display > Views...*) as shown below.

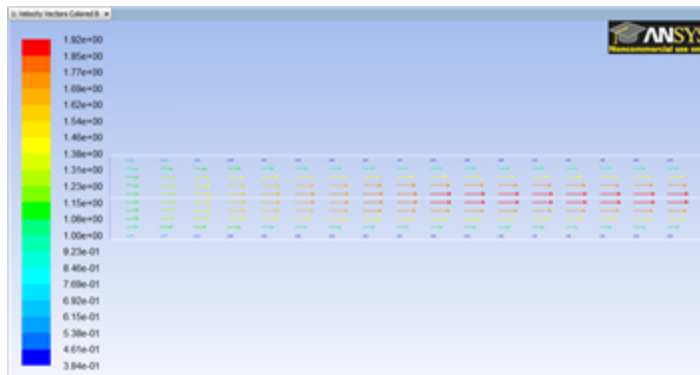


[Higher Resolution Image](#)

Under **Mirror Planes**, only the **axis (or centerline)** surface is listed since that is the only symmetry boundary in the present case. Select **axis (or centerline)** and click **Apply**, as shown below.



Then click **Close** to exit the **Views** menu. Your vector field should have been reflected across the axially axis as, shown below.

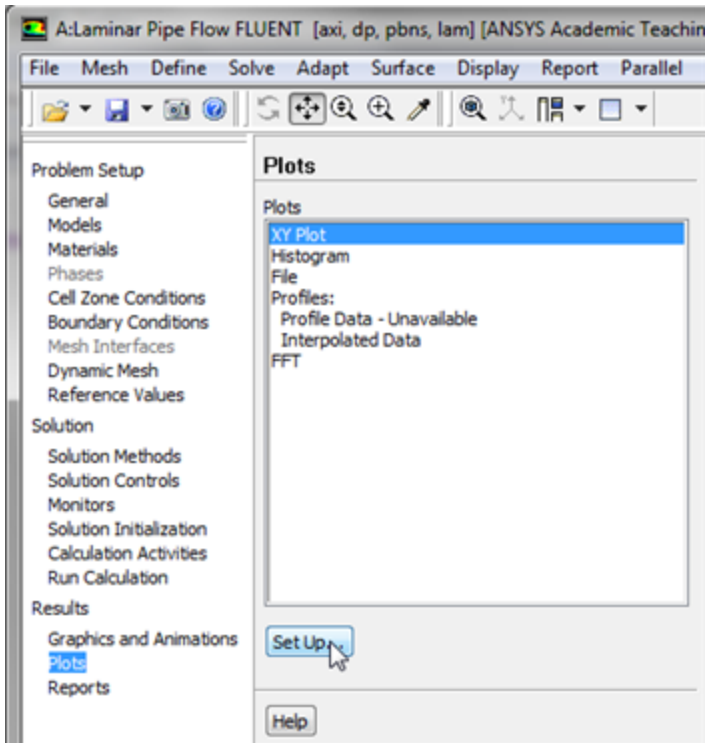


[Higher Resolution Image](#)

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.

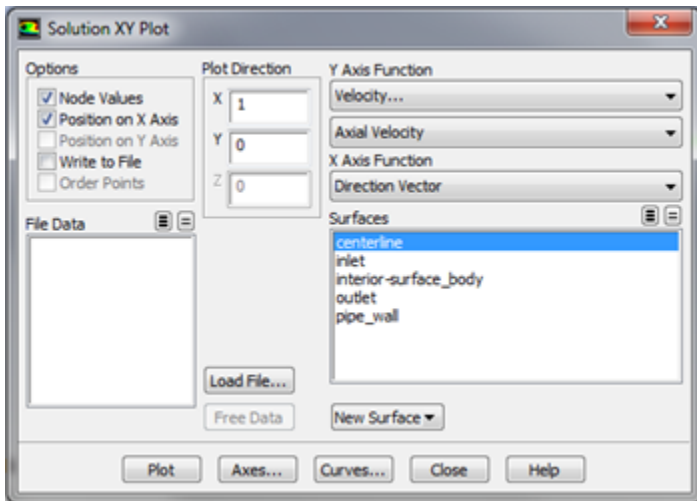
Centerline Velocity

Here, we'll plot the variation of the axial velocity along the centerline. In order to start the process (**Click** *Results > Plots > XY Plot... > Set Up..*) as shown below.



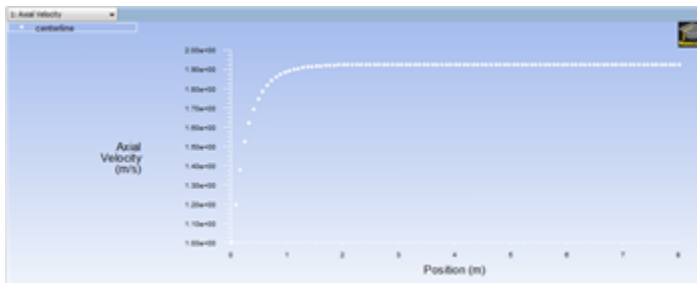
[Higher Resolution Image](#)

In the **Solution XY Plot** menu make sure that **Position on X Axis** is selected, and **X** is set to 1 and **Y** is set to 0. This tells FLUENT to plot the x-coordinate value on the abscissa of the graph. Next, select **Velocity...** for the first box underneath **Y Axis Function** and select **Axial Velocity** for the second box. 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. Your **Solution XY Plot** should look exactly the same as the following image.



[Higher Resolution Image](#)

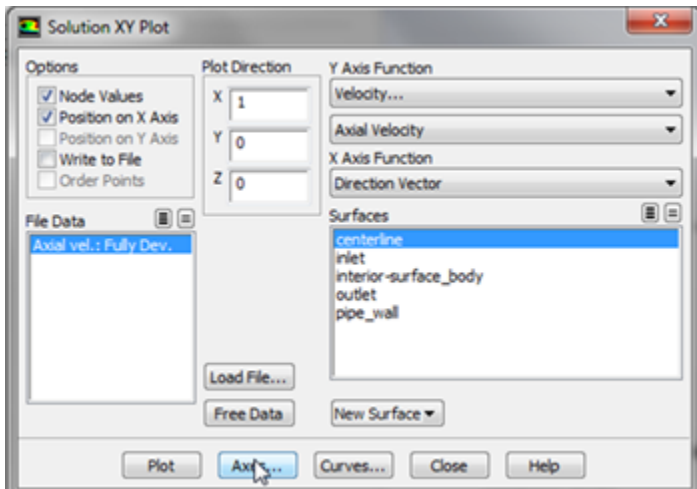
Now, click **Plot**. The plot of the axial velocity as a function of distance along the centerline now appears.



[Higher Resolution Image](#)

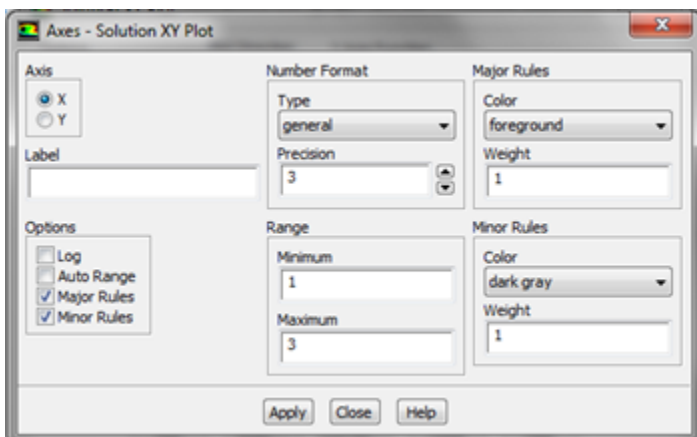
In the graph that comes up, we can see that the velocity reaches a constant value beyond a certain distance from the inlet. This is the fully-developed flow region. At this point the graph will be modified such that the fully developed regions results are truncated. That is, the range of the axes will be

reconfigured. Open the **Solution XY Plot** menu, then click on **Axes...**, as shown below.



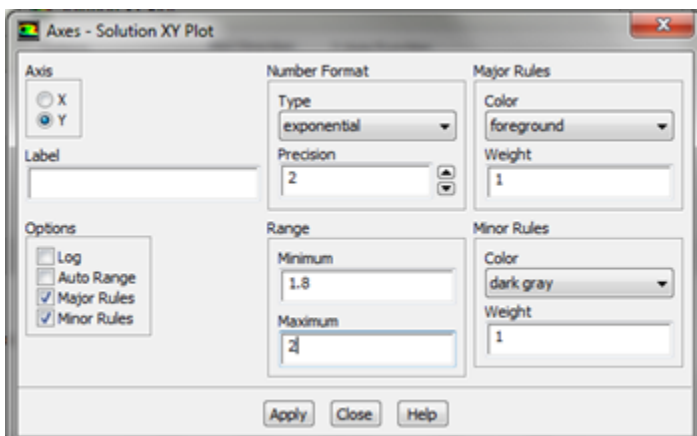
[Higher Resolution Image](#)

Then, deselect **Auto Range**, which is located under **Options**. The boxes under **Range** should now be accessible. Next, select **X**, which is located under **Ax** **is**. Enter 1 for **Minimum** and 3 for **Maximum** under **Range**. At this point the grid lines will be turned on in order to help estimate where the flow becomes fully developed. Check the boxes next to **Major Rules** and **Minor Rules** under **Options**. At this point your **Axes - Solution XY Plot** menu should look exactly the same as the image below.



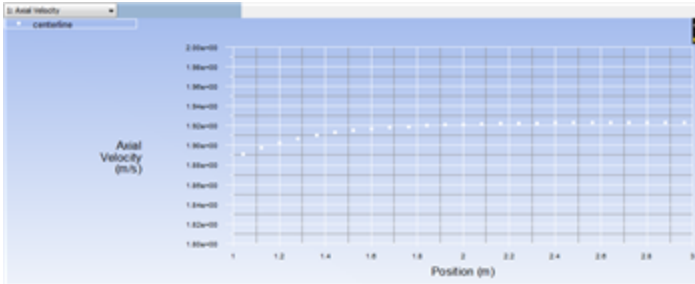
[Higher Resolution Image](#)

Lastly, click **Apply**. Now, that the X axis has been formatted, we will move on to formatting the Y axis. Select **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. At this point your **Axes - Solution XY Plot** menu should look exactly the same as the image below.



[Higher Resolution Image](#)

We have now finished specifying the range for each axis, so click **Apply** and then **Close**. At this point, the graph can be replotted. Go to the **Solution XY Plot** menu and click **Plot** to plot the graph again with the new axes extents.



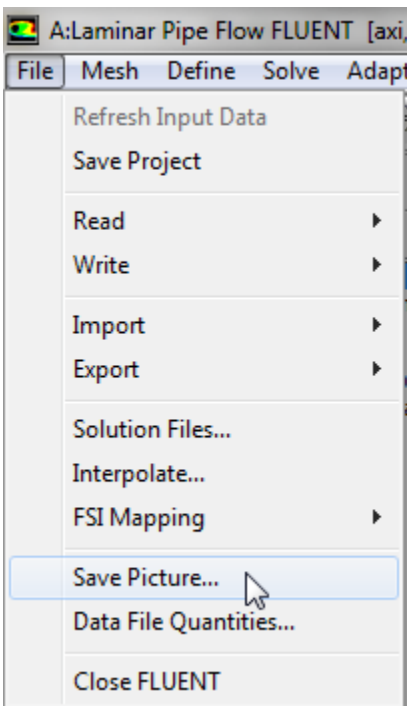
Higher Resolution Image

From the image above, one 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 .

Saving the Plot

In this section, we will save the data from the plot and a picture of the plot. The data from the plot will be saved first. In order to save the plot data open the **Solution XY Plot** menu and then select **Write to File**, which is located under **Options**. The **Plot** button should have changed to **Write....** Click on **Write....**, and then enter `vel.xy` as the XY File Name. Next, click **OK**. Check that this file has been created in your FLUENT working directory. Lastly, close the **Solution XY Plot** menu.

At this point, we'll save a picture of the plot. First click on **File** then click **Save Picture**, as shown below.



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.

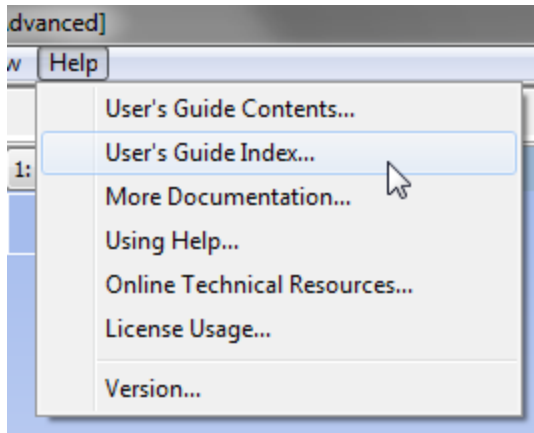
After selecting your desired image format and associated options, click on **Save...**

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. In order to access the online help first (**click**) **Help > Users Guide Index** as shown in the following image.



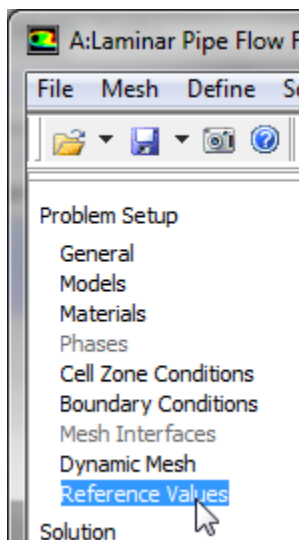
Click on S in the links on top and scroll down to **skin friction coefficient**. Click on the first link (normally, you would have to go through each of the links until you find what you are looking for). There you can see the following excerpt on the skin friction coefficient as well as the equation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

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

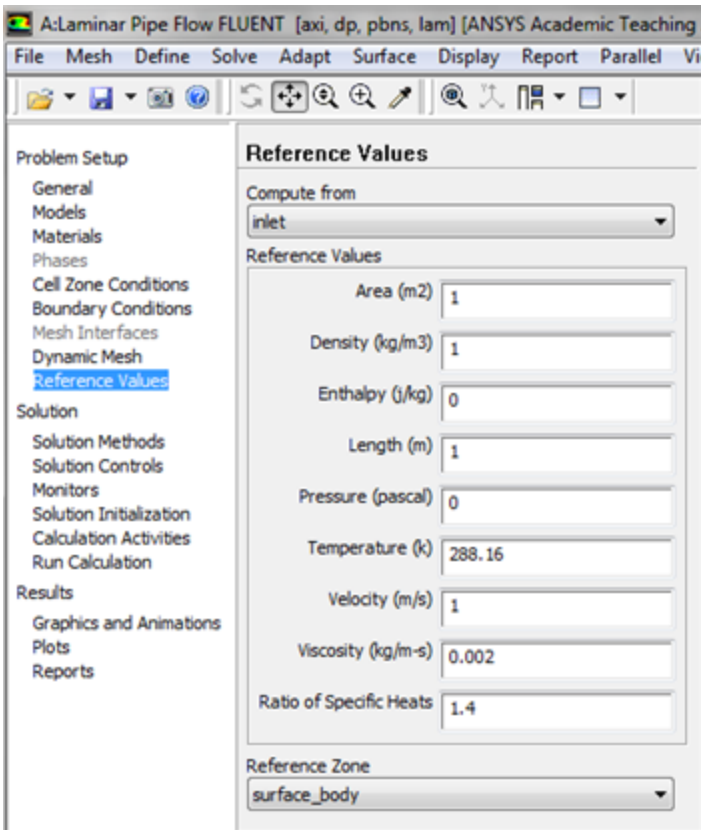
$$C_f \equiv \frac{\tau_w}{\frac{1}{2}\rho_{\text{ref}}v_{\text{ref}}^2} \quad (31.4-36)$$

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](#) task page. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

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

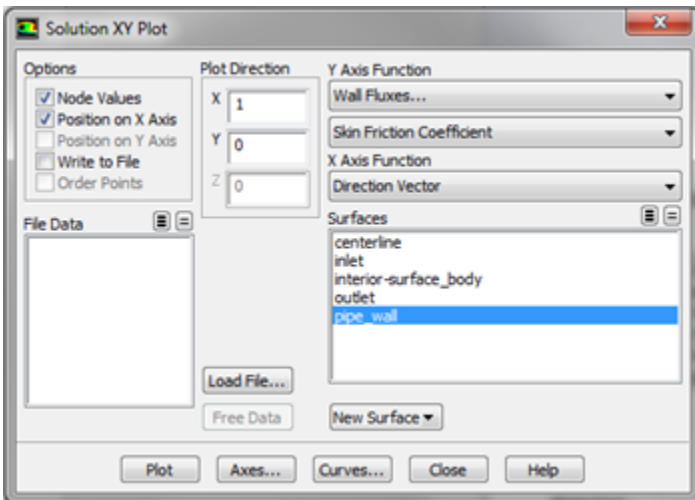


Then, set **Compute From** to **inlet**, to tell FLUENT to calculate the reference values from the values at inlet. Check that density is 1 kg/m^3 and velocity is 1 m/s . (Alternately, you could have just typed in the appropriate values). Your **Reference Values** should look the same as the following screen snapshot.



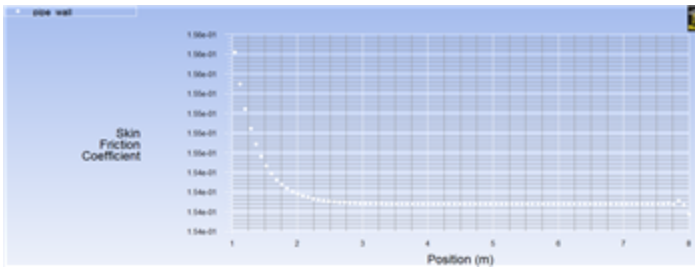
[Higher Resolution Image](#)

Now, reopen the **Solution XY Plot** menu. Uncheck the **Write to File** check box under **Options**, since we want to plot to the window. The **Options** and **Plot Direction** can be left 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**, only select **pipe_wall**. At this point, your **Solution XY Plot** menu should look exactly like the following image.



[Higher Resolution Image](#)

Now, the ranges of each axis will be specified. Click on **Axes...** within the **Solution XY Plot** menu 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 deselect **Range Auto-Range** first if it is checked). Click **Apply**, then click **Close**. Lastly, click **Plot** in the **Solution XY Plot** menu. You should obtain the following plot.



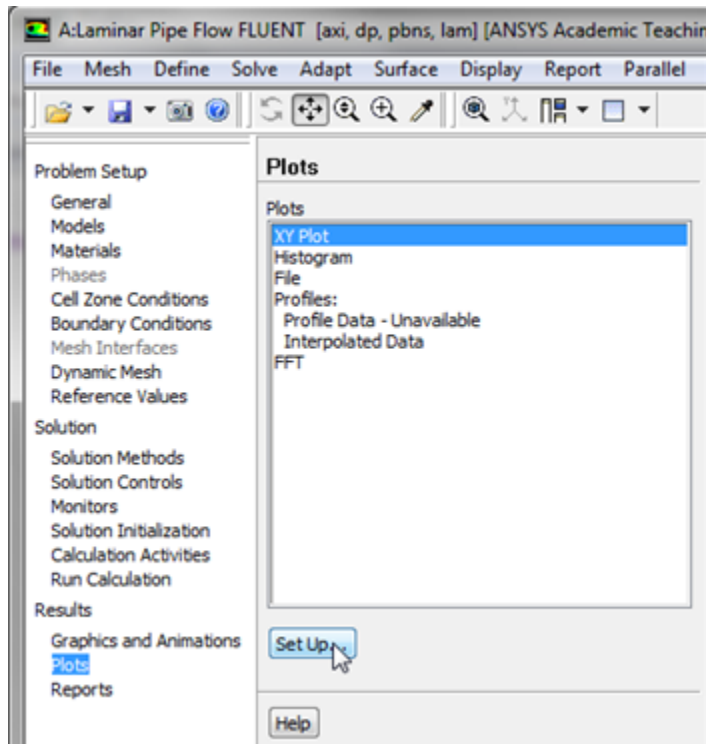
[Higher Resolution Image](#)

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 1.54×10^{-5} .

In order to save the data from this plot, first reopen the **Solution XY Plot** menu. Then, select **Write to File** under **Options** and click **Write....** Enter `cf.xy` for **XY File** and click **OK**.

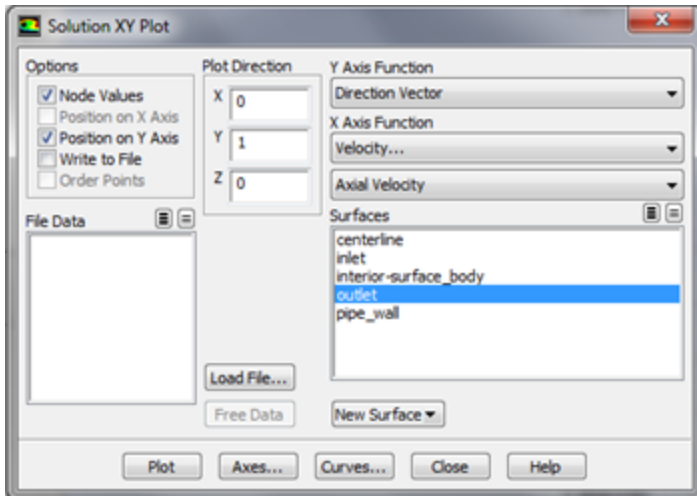
Velocity Profile

In this section we will plot the velocity at the outlet as a function of the distance from the center of the pipe. In order to start the process **(Click) Results > Plots > XY Plot... > Set Up..** as shown below.



[Higher Resolution Image](#)

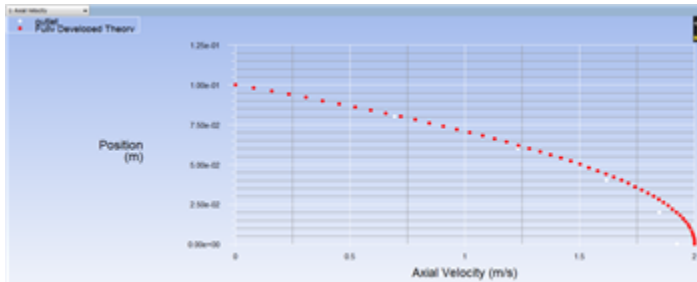
For this graph, the y axis of the graph will have to be set to the y axis of the pipe (radial direction). To plot the position variable on the y axis of the graph, uncheck **Position on X Axis** under **Options** and choose **Position on Y Axis** instead. To make the position variable the radial distance 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, select **Velocity...** for the first box underneath **X Axis Function**, and select **Axial Velocity** for the second box. Next, select **outlet**, which is located under **Surfaces**. Then, uncheck the **Write to File** check box under **Options**, so the graph will plot. Your **Solution XY Plot**, should look exactly like the image below.



Higher Resolution Image

Next, click on **Axes** in the **Solution XY Plot** menu. Then, change both the x and y axes to **Auto-Range**. (Don't forget to click apply before selecting a different axis). Click **Close** in the **Axes - Solution XY Plot** menu.

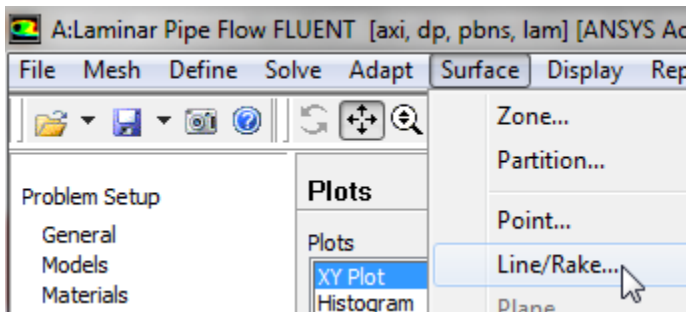
It is of interest to compare the velocity profile with the theoretical parabolic profile. In order to plot the theoretical results, first click [here](#) to download the necessary file. Save the file to your working directory. Next, go to the **Solution XY Plot** menu and click **Load File...** and select the file that you just downloaded, *profile_fdev.xy*. Lastly, click **Plot** in the **Solution XY Plot** menu. You should then obtain the following figure.



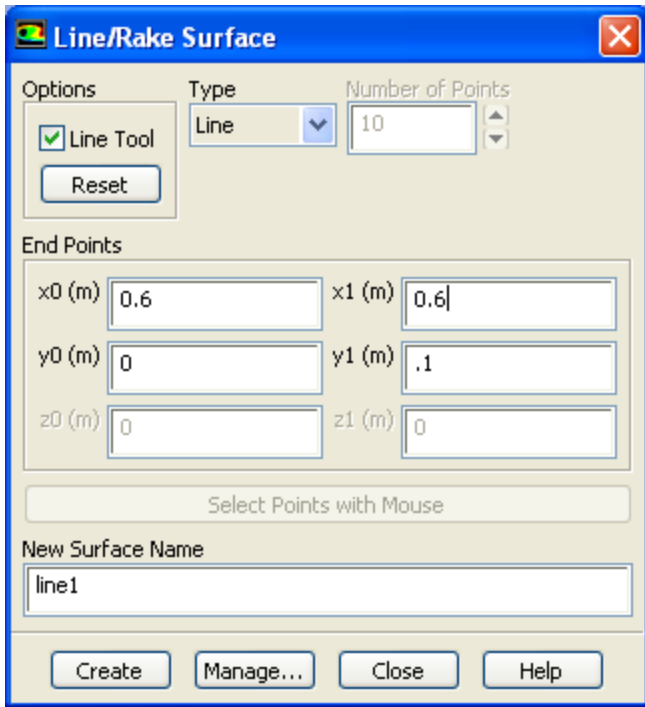
Higher Resolution Image

Notice, how results compare relatively well with the theoretical parabolic profile. In order to save the data from this plot, first reopen the **Solution XY Plot** menu. Then, select **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, we will add profiles at $x=0.6\text{m}$ ($x/D=3$) and $x=0.12\text{m}$ ($x/D=6$) to the previous plot. In order to create the profiles, we must first create vertical lines using the **Line/Rake** tool. First, **(Click) Surface < Line/Rake** as shown in the following image.



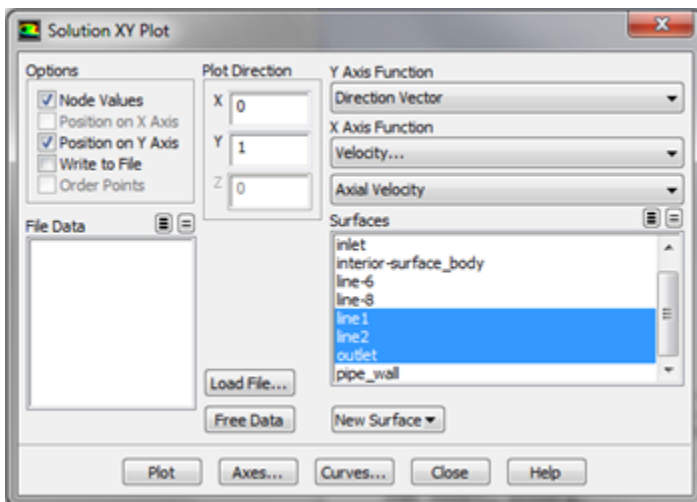
We'll create a straight line from $(x_0, y_0)=(0.6, 0)$ to $(x_1, y_1)=(0.6, 0.1)$. Select **Line Tool** under **Options**. Enter $x_0=0.6$, $y_0=0$, $x_1=0.6$, $y_1=0.1$. Enter *line1* under **New Surface Name**. Click **Create**.



To see the line that you just created, (Click) **Display > Mesh**. 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.) The white vertical line appearing to the right is **line1**, as shown in the image below.

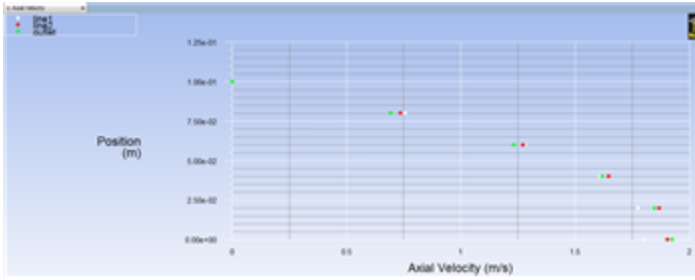


Similarly, create a vertical line called **line2** at $x=1.2$; $(x_0, y_0)=(1.2, 0)$ to $(x_1, y_1)=(1.2, 0.1)$. Display it in the graphics window to check that it has been created correctly. Now, we can plot the velocity profiles at $x=0.6\text{m}$ ($x/D=3$) and $x=0.12\text{m}$ ($x/D=6$) along with the outlet profile. First, open the **Solution XY Plot** menu. Under **Surfaces**, in addition to **outlet**, select **line1** and **line2**. Make sure **Node Values** is selected under **Options**. Now, your **Solution XY Plot** menu should look exactly like the following image.



[Higher Resolution Image](#)

Lastly, click **Plot** and you should obtain the following output. 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 under predicts 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.

[Go to Step 7: Verification & validation](#)

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

[Go to all FLUENT Learning Modules](#)