

Appendix J

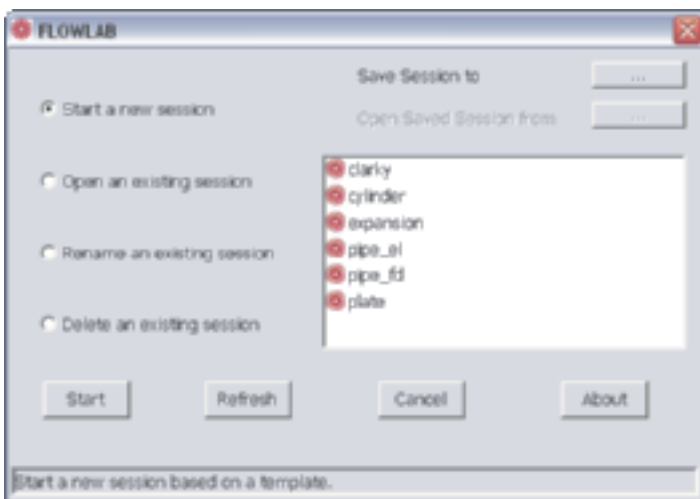
FlowLab Tutorial and User's Guide

FlowLab Basics (*FlowLab ver. 1.2*)

The following tutorial takes you through a step-by-step procedure to simulate developing flow in a pipe, along with relevant notes where needed. The procedures for using all the FlowLab templates is quite similar, so going through this tutorial will prepare you to use the other templates. Other sources of information include individual descriptions for each template and the FlowLab User's Guide (see the textbook's [main FlowLab page](#)).

Developing Flow in a Pipe

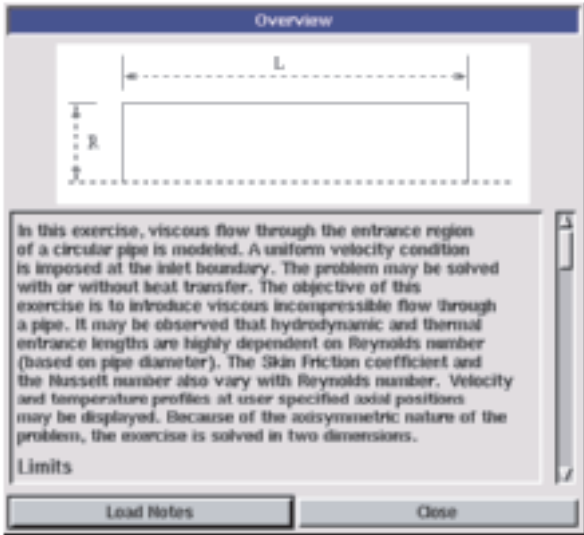
Preliminary: Start FlowLab with the developing pipe flow template: pipe_el. Double-click on the FlowLab icon to start the program. You will be presented with a pop-up window:



Click on “Start a new session,” select the pipe_el template (i.e., entrance length of pipe), and click on the **Start** button. FlowLab will open with this template loaded. The following steps take you through the problem setup, running the simulation, and examining the results. In general, the solution to the problem will progress from left to right in the “Operation” tool bar shown below:

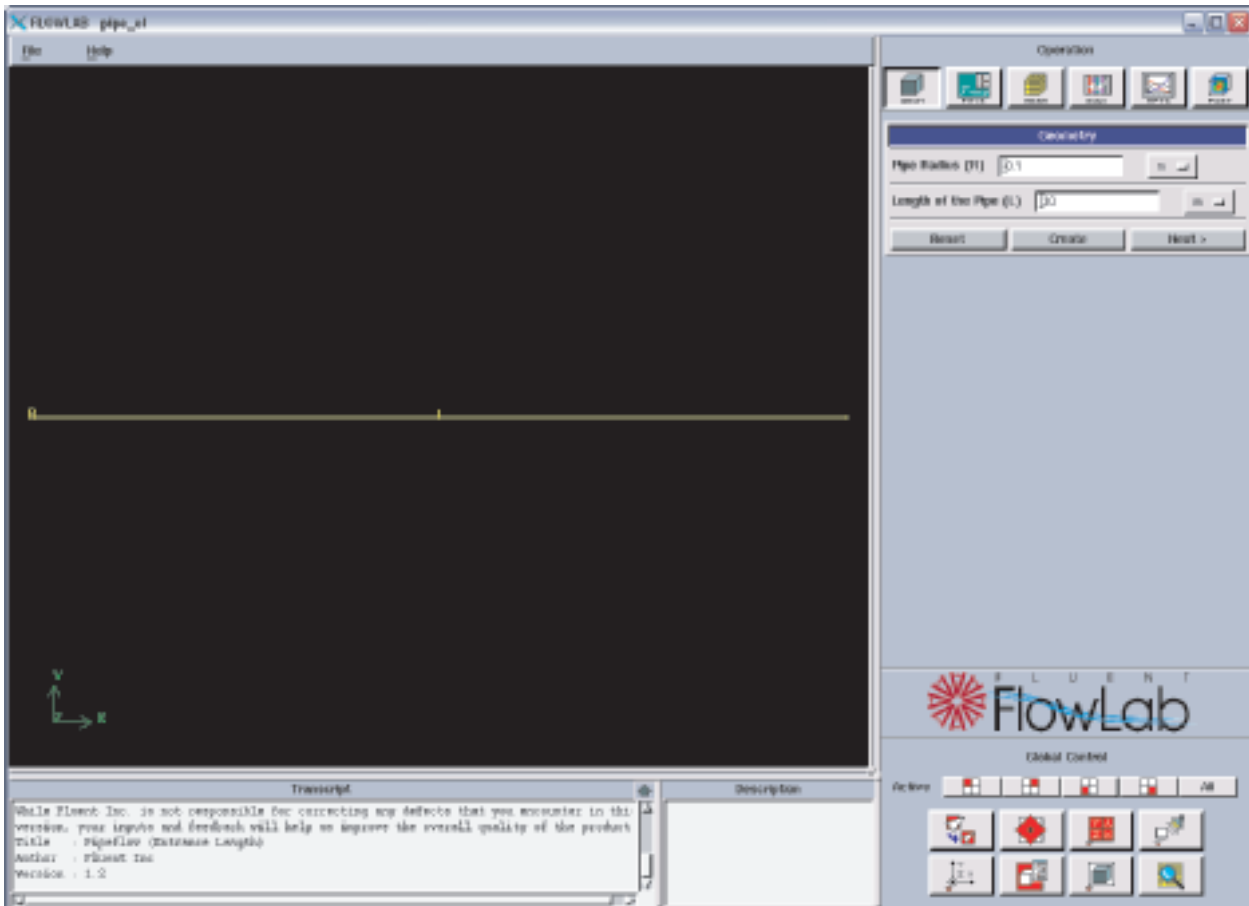


When you first start up the template, a pop-up window (shown below) will appear that gives an overview of the problem involved with this template including input and output information. It is advisable to read through this overview if it is the first time using a particular template.



If you close this window, you can reopen it by selecting **File** → **Problem Overview** from the pull-down menu.

Step 1. Create Geometry: Click on the geometry [GEOM] button in the Operation tool pad to open the Geometry Window, as shown below. Enter values for pipe radius and length of the pipe. If you have questions on maximum and



minimum allowed values, look through the **Problem Overview** described above. For this example, the pipe radius is set to 0.1 m and the pipe length to 30 m. Click on the **Create** button to create the geometry in the graphics window. Each time you make changes to the geometry, you will need to re-create the geometry by clicking on the **Create** button.

Once the geometry has been created, go on to the next step of specifying the physics of the problem. You can do this either by clicking on the **Next** button in the Geometry window or clicking on the physics [PHYS] button in the Operation tool pad.

Step 2. Physics: In this step, the user will select whether the flow is laminar or turbulent, whether to include heat transfer in the problem, the boundary conditions and materials that help describe the specific physics of the problem. For this example, the type of flow is set to laminar, the heat transfer has been turned off and the boundary condition and materials were selected to produce a Reynolds number of approximately 450, as shown below. Click on the **Boundary Condition** and **Materials** buttons to open the respective windows. Notice that certain parameters related to the heat transfer are grayed out because it was decided not to include this in the model.

The image shows two side-by-side screenshots of the FlowLab software interface. The left window is titled "Physics" and contains sections for "Viscous Condition" (with "Laminar" selected), "Heat Transfer" (with "Boundary Condition" selected), and "Materials" (with "Re # 452.101" and a "Compute" button). The right window is titled "Boundary Condition" and contains fields for "Inlet Velocity" (0.0303 m/s), "Inlet Temperature" (298 K), "Wall Thermal Condition" (with "Heat Flux" selected), "Heat Flux" (0 W/m²), and "Temperature" (298 K). Both windows have "Reset" and "OK" buttons at the bottom.

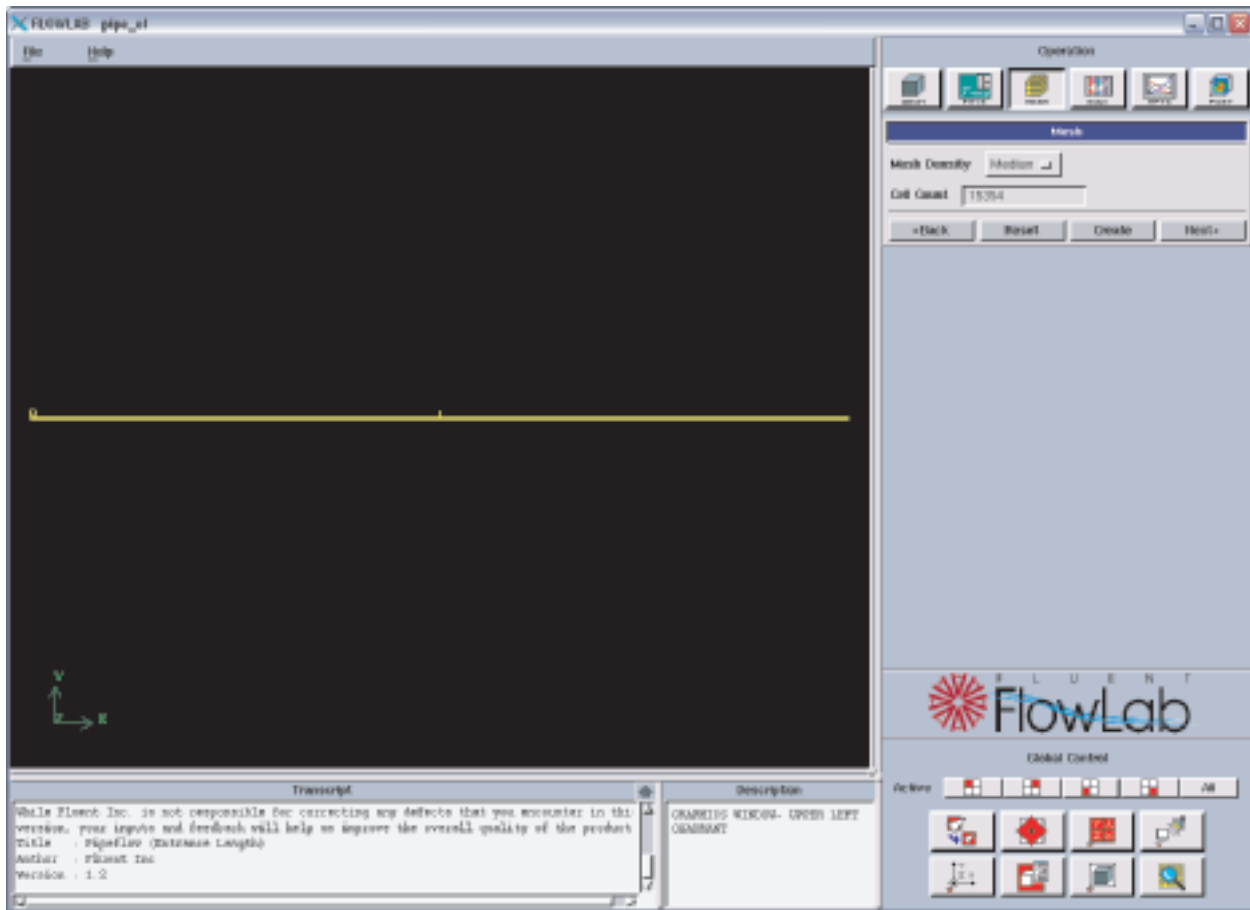
The image shows a screenshot of the "Materials" window in FlowLab. It contains fields for "Density" (1225 kg/m³), "Viscosity" (5.642e-05 kg/m-s), "Specific Heat" (500 J/kg-K), and "Thermal Conductivity" (0.025 W/m-K). The window has "Reset" and "OK" buttons at the bottom.

As you alter the inlet velocity, density, or viscosity, you can click on the **Compute** button to have FlowLab update the Reynolds number. Note that the minimum inlet velocity is 0.01 m/s according to the **Problem Overview**. Once complete, click on the **OK** button in the Materials and Boundary Condition windows to close them. The physics for the problem has now been set. The next step is the creation of the mesh. Either click the **Next** button in the Physics window or click on the mesh [MESH] button in the **Operation** tool pad.

Step 3. Mesh: In this step, you will select the grid resolution that you would like the flow solver to use for this particular problem. At the **Mesh Density**, you can activate a pull-down menu to select between **Coarse**, **Medium**, and **Fine** meshes. For this problem, a **Medium** mesh was selected for the grid, as shown below.

The image shows a screenshot of the "Mesh" window in FlowLab. It contains a "Mesh Density" pull-down menu set to "Medium" and a "Cell Count" field showing "15354". The window has "Back", "Reset", "Create", and "Next" buttons at the bottom.

Once you have selected the mesh density, click on the **Create** button to generate the grid. A progress bar at the top of the graphics window will signify that FlowLab is working to generate the grid. Once complete, it will be displayed in the graphics window as shown in the following figure. Note that changes to the geometry will not only require you to recreate the geometry, but also to create a new mesh.



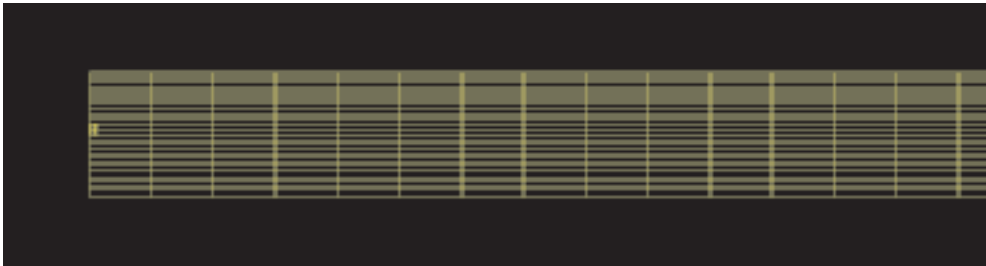
You can zoom into a portion of the geometry/grid in the graphics window. The following mouse commands are used in the graphics window (see the [Commonly Used Commands Summary](#) for more information):

- Left mouse button: used to rotate geometry in three dimensions
- Middle: used to translate the geometry
- Right: used to zoom in and out – drag down to zoom in and drag up to zoom out

Other notes:

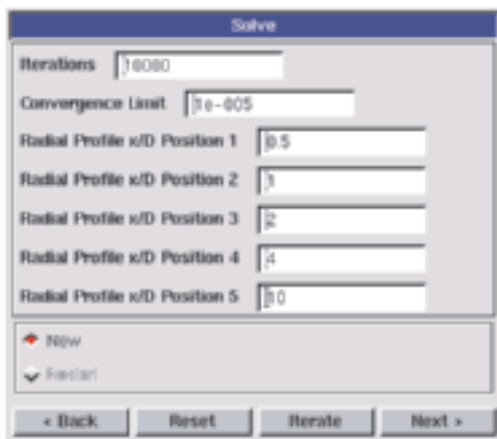
1. If you rotated the object and want to return to the standard X-Y view, click on the **Orient Model** button  in the **Global Control** tool pad at the bottom right of the FlowLab window.
2. You can also zoom into a particular region by holding down the **CTRL** button and using the left mouse button to drag open a rectangle in the area of interest.
3. After zooming into a particular region, if you want to display the full geometry again, click on the **Fit To Window** button  in the **Global Control** tool pad.

If you zoom into the entrance region, the medium grid will look as follows:



Notice the stretched grid near the wall and the fact that this problem is being solved in an axisymmetric fashion. The next step is to solve the flow field – click **Next** or the solve [SOLV] button.

Step 4. Solve: In this step, you will set up the solution parameters and start the iteration process. You will set the maximum number of iterations, the convergence limit, and pipe locations for radial profiles to be plotted. For this example, the default Iteration number of 10,000 and the convergence limit of 1.0×10^{-5} are used. Five radial profile positions have been selected as shown below. For more information on these settings, see the individual [template description](#) or the [FlowLab User's Guide](#).

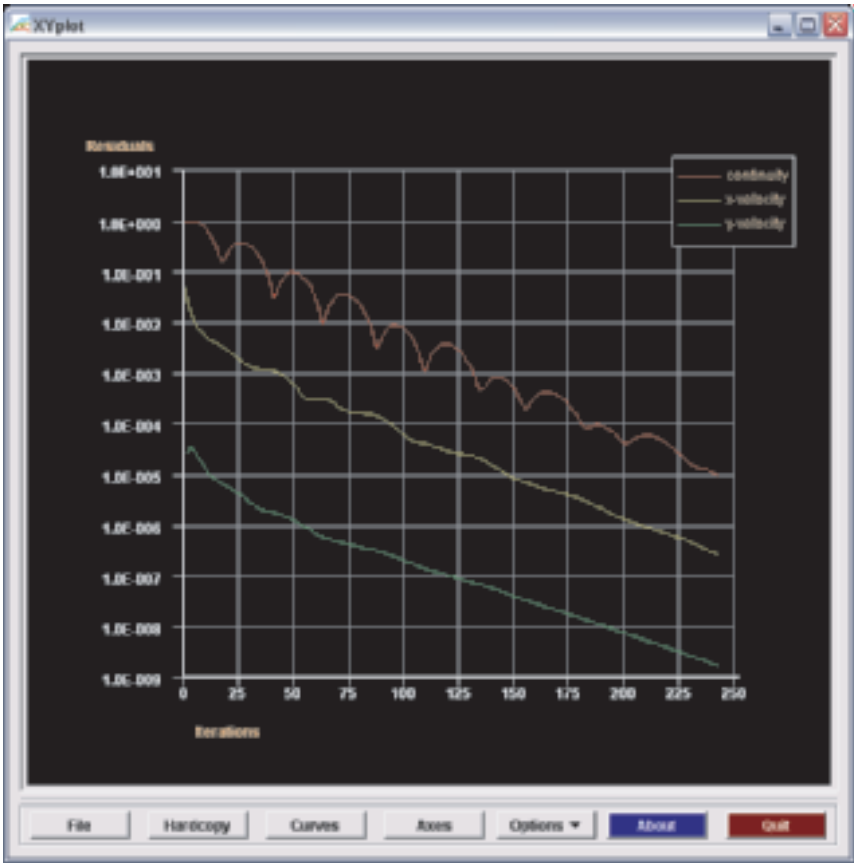


Once you have selected the values, you are ready to start the iteration process. Make sure the “New” is checked since this will be a new solution; then click on the **Iterate** button. FlowLab will show a progress meter at the top of the graphics window to indicate it is starting the iteration process. An *xyplot* window will open to display the **Residuals** (see [FlowLab User's Guide](#) for more information) from the iterations – see the following page for the Residual plot from the above calculation.

As the Residual plot shows, this simulation only took approximately 240 iterations to reach a solution. FlowLab will alert you when the solution has been reached, i.e., all the residuals have reached the value set for the convergence limit. NOTE: this problem converges rather quickly. Depending on the problem setup, some simulations will take longer to converge to a solution. The student is advised to read through the individual descriptions for each template prior to running simulations.

During the run, there is an **Interrupt** button available at the top of the graphics window if you need to interrupt the simulation. The **Restart** button in the Solve window can be used to restart a simulation if it was interrupted or if you did not reach convergence in the number of iterations allowed.

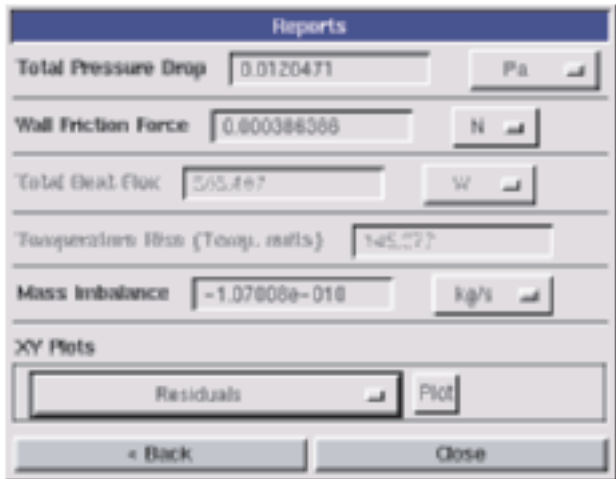
Now it is time to look at the results of your simulation – click **Next** or the reports [RPTS] button to view a report of the flow simulation.



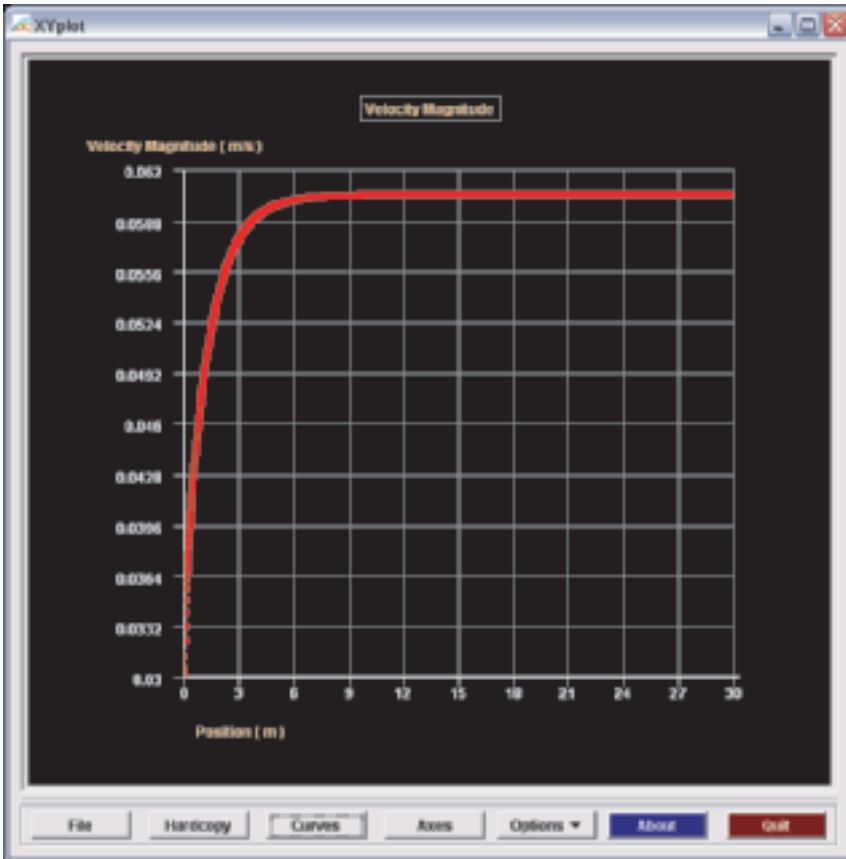
Step 5. Reports: The **Reports** section of FlowLab provides some quantitative data from the simulation. For this template, the **Reports** provides total pressure drop, wall friction force, and multiple *xy* plots. For contour and vector plots, the postprocessing [POST] component will be used, which is covered in the next step.

The Reports window for this run is shown below. Heat flux and temperature rise are grayed out because the heat transfer option was not selected in the Physics section.

Note: If you want to show the plot of **Residuals** again after the simulation is complete, select this option under the **XY Plots** from the **Reports** window and click the **Plot** button.



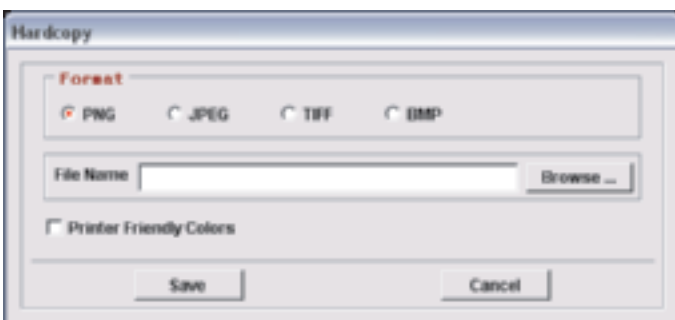
The first problem from Chapter 8 asks the student to explain some of the trends in the centerline velocity profile. To plot this velocity, select the **Centerline Velocity Distribution** from the pull-down menu of the **XY Plots** – then click on the **Plot** button. An *xyplot* window will be opened showing the selected plot. A sample is shown below.



For better viewing, the legend was turned off under the **Options** pull-down menu in the plot window.

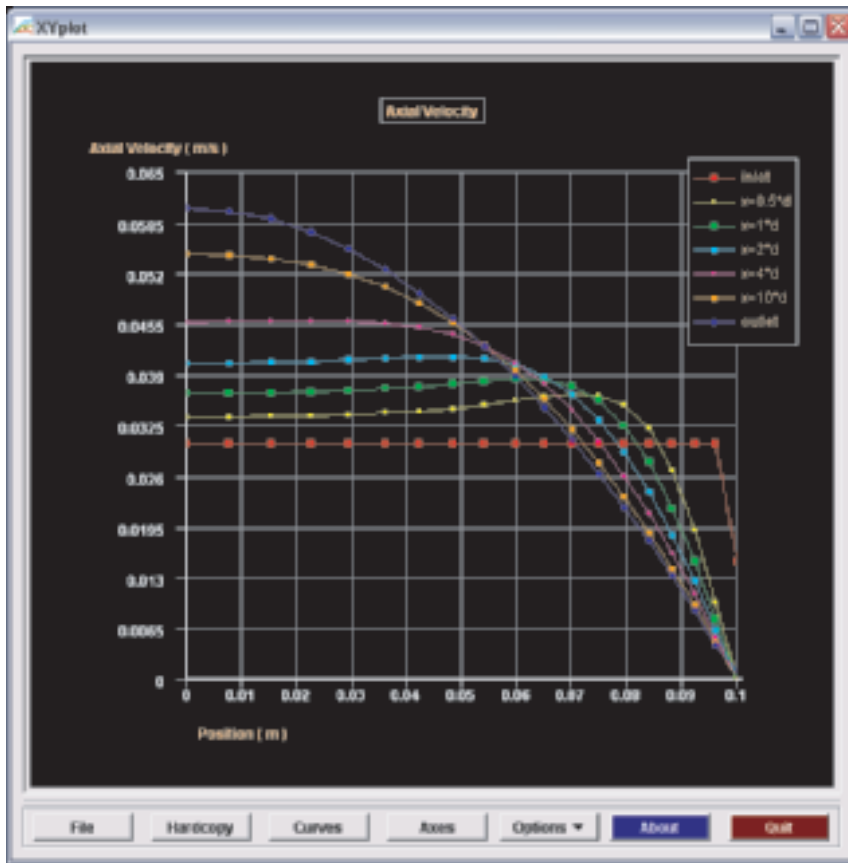
Note 1: One available option in the *xyplot* window is export data. This allows the user to output the plot data in ascii format so that they can be imported into other plotting packages such as Excel. To export the data, click on the **File** button at the bottom of the window and make sure to select “Export data.” This comes in handy when making direct comparisons between different runs. See the [Commonly Used Command Summary](#) for more information.

Note 2: The other method for documenting the *xy* plot results is to use the hardcopy feature of the *xyplot* window. Click on the **Hardcopy** button at the bottom and you will be presented with the following window:



With this feature, you can output your results to an image file in various formats.

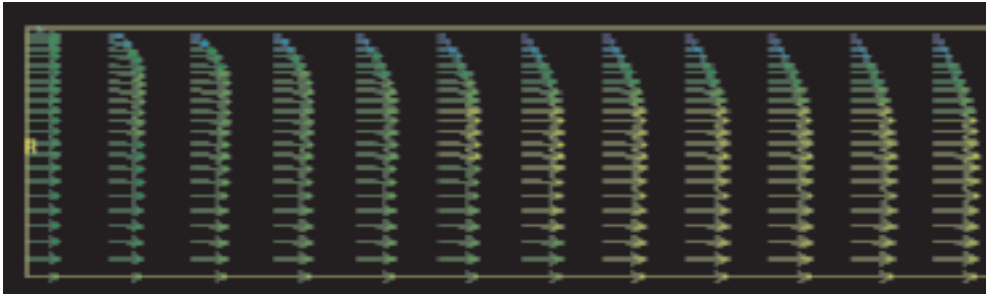
Part (c) of the problem asks the student to look at the differences in the velocity profile between the entrance and the fully developed region. Make use of the radial profile selections that were made when setting up the iteration process. Select the **Profiles of Axial Velocity** from the pull-down menu on the **XY Plots**. The results for this example are shown below. The different plots correspond to the radial profile selections made previously. The plot automatically includes the inlet and outlet profiles.



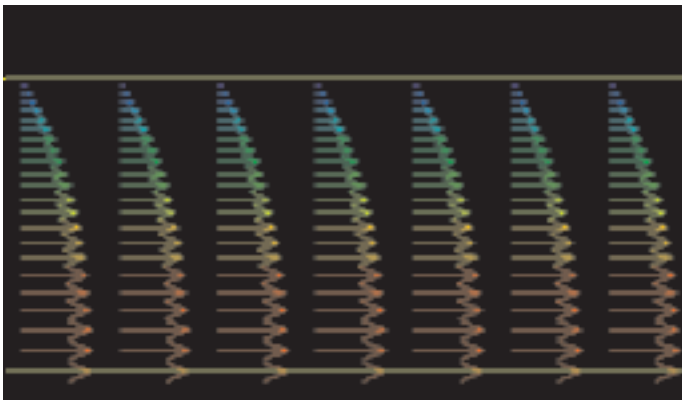
As the plot shows, the position of x/D equal to 10 is approaching the outlet profile.

Step 6. Post-Process Data: For some problems, it will be enough for the students to make use of data from the reports [RPTS] section of FlowLab. For postprocessing of the data such as contour plots and vector plots, the last button in the **Operations** tool pad needs to be used – [POST]. For example, to plot out the velocity vectors for this solution, select **vector** from the **Postprocessing Objects** and click on the **Activate** button. The velocity vectors will appear in the graphics window.

With the velocity vectors plotted on the geometry, you can zoom into different regions (as described above) of the pipe to inspect the velocity profiles. If you are asked to visualize the inviscid core of the entrance region, examining the velocity vectors is one method. For this example the snapshot on the following page shows the velocity vectors in the entrance region of the pipe. Notice the uniform flow region in the center of the pipe.



In the fully developed region of the pipe, the velocity profiles look like:



Using the postprocessing section of FlowLab to view results is a bit more complicated than the **Reports** section. Some basic notes are listed below to help in examining some of the results. See the individual template descriptions and the [FlowLab User's Guide](#) for more help.

1. When you are done with one type of visualization, you can turn it off by clicking on the **Deactivate** button (unless you want to overlay two plots).
2. You can overlay results such as contours and vectors by activating both of these objects.
3. Results plotted in the graphics window can be saved in several different ways:
 - a. Select **File** → **Print Graphics** to print the graphics window contents to either a printer or a file
 - b. Right-click on the FlowLab title bar and select the **Edit** menu item. This allows you to copy a selected rectangle or "all" to the clipboard, file, or printer.
4. You can change the parameters of the objects by clicking on the **Modify** button. This allows you to customize the look of the vectors or the contours. For example, in the velocity vector plots above, the scale of the vectors was changed to 5 for easier viewing. Click on **Modify** to open the **Modify Simulation Object** window. Then click the **Edit** button in the velocity vectors section to open the **Specify Vector Attributes** window. See page 10 for a sample of windows.

Step 7. Save Your Project: To save the current simulation, select **File** → **Save** or **File** → **Save As** from the pull-down menu to save FlowLab information to the hard disk. This should be done throughout the problem solution. In the **Save** pop-up window, enter a representative file name. FlowLab will create a run directory with this name that contains all relevant files including problem setup and results. The directory is written to the "myflowlab" directory. Click the **Browse** button if you are unsure where this is located.

