CFD and Software

Numerical simulation with Fluent 6.3.26: ”Project Steps”

Important! this document was inspired by the Gambit 2.4.6 and Fluent 6.3.26 User Guide and presented here to students after being well reorganized and structured from a pedagogical point of view.

This part concerns setup, calculations and displaying results with the software Fluent 6.3.26.

Step 1: Set Up Problem in FLUENT

Launch Fluent 6.3

Lab Apps > FLUENT 6.3.26

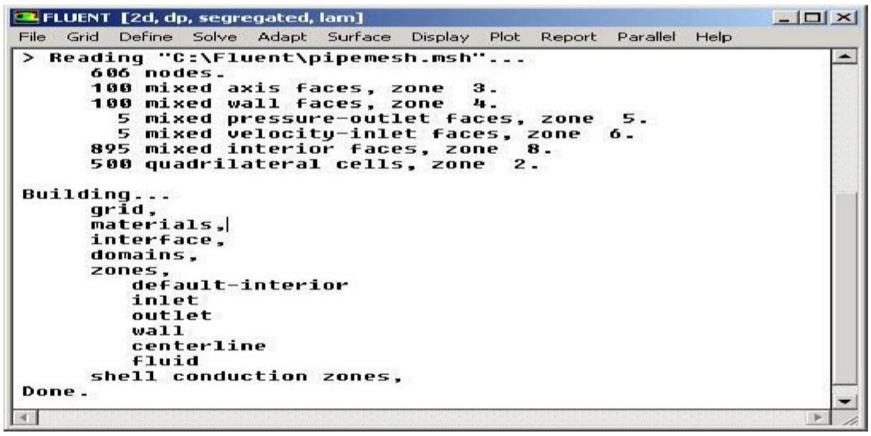
Select "**2ddp**" (double precision) from the list of options and click Run.

The "**2ddp**" option is used to select the 2-dimensional, double-precision solver. In the double-precision solver, each floating point number is represented using 64 bits in contrast to the single-precision solver which uses 32 bits. The extra bits increase not only the precision but also the range of magnitudes that can be represented. The downside of using double precision is that it requires more memory.

Import Grid

Main Menu > File > Read > Case...

Navigate to the working directory and select the *pipe.msh* file. This mesh file was created using the preprocessor *GAMBIT* in the previous step. FLUENT reports the mesh statistics as it reads in the mesh:




```
> Reading "C:\Fluent\pipemesh.msh"...
      606 nodes.
      100 mixed axis faces, zone  3.
      100 mixed wall faces, zone  4.
        5 mixed pressure-outlet faces, zone  5.
        5 mixed velocity-inlet faces, zone  6.
      895 mixed interior faces, zone  8.
      500 quadrilateral cells, zone  2.

Building...
      grid,
      materials,
      interface,
      domains,
      zones,
         default-interior
         inlet
         outlet
         wall
         centerline
         fluid
      shell conduction zones,
Done.
```

Figure 1. Window of text commands

Check the number of nodes, faces (of different types) and cells. There are 500 quadrilateral cells in this case. This is what we expect since we used 20 divisions in the radial direction and 100 divisions in the axial direction while generating the grid. So the total number of cells is 5×100 = 500 (10×1000 = 500 and 21×100 = 2100).
Also, look under zones. We can see the four zones inlet, outlet, wall, and centerline that we defined in *GAMBIT*.

Check and Display Grid

First, we check the grid to make sure that there are no errors.

Main Menu > Grid > Check

Any errors in the grid would be reported at this time. Check the output and make sure that there are no errors reported. Check the grid size:

Main Menu > Grid > Info > Size

The following statistics should appear:

```
Grid Size

Level    Cells    Faces    Nodes   Partitions
    0      500     1105      606            1

 1 cell zone, 5 face zones.
```

Figure 2. Grid informations

Display the grid:

Main Menu > Display > Grid...

Make sure all 5 items under Surfaces is selected. Then click Display. The graphics window opens and the grid is displayed in it. You can now click Close in the *Grid Display* menu to get back some desktop space. The graphics window will remain.

Some of the operations available in the graphics window are:

Translation: The grid can be translated in any direction by holding down the Left Mouse Button and then moving the mouse in the desired direction.

Zoom In: Hold down the Middle Mouse Button and drag a box from the Upper Left Hand Corner to the Lower Right Hand Corner over the area you want to zoom in on.

Zoom Out: Hold down the Middle Mouse Button and drag a box anywhere from the Lower Right Hand Corner to the Upper Left Hand Corner.

Use these operations to zoom into the grid to obtain the view shown below.

➢ Note! The zooming operations cannot be performed without a middle mouse button.

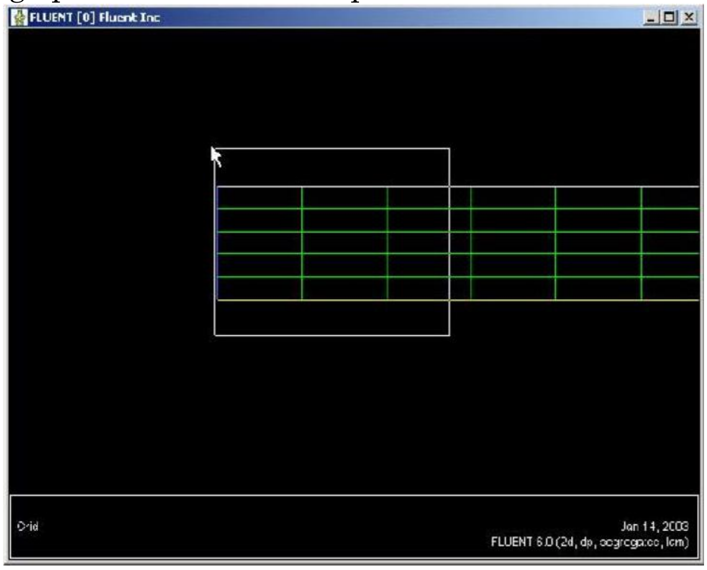


Figure 3. Zoom view

You can also look at specific parts of the grid by choosing the boundaries you wish to view under Surfaces (click to select and click again to deselect a specific boundary). Click Display again when you have selected your boundaries. For example, the wall, outlet, and centerline boundaries have been selected in the following view:

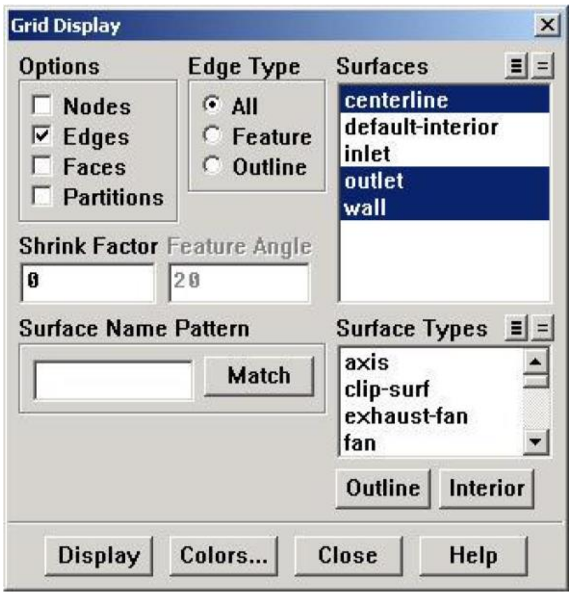


Figure 4. Grid display box

These options will display the graph:

Figure 5. Display window

For convenience, deselects all of the boundaries at button next to Surfaces selects all of the boundaries while the .

Close the *Grid Display Window* when you are done.

Define Solver Properties

Main Menu > Define > Models > Solver

Choose Axisymmetric under Space. We will use the defaults of pressure based ("segregated", in older versions) solver, implicit formulation, steady flow and absolute velocity formulation. Click OK.

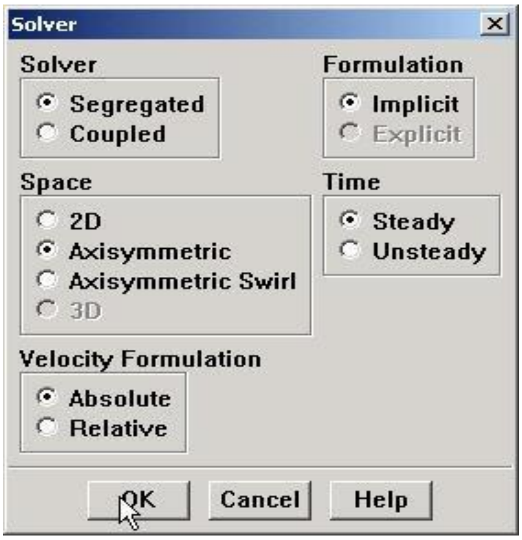


Figure 6. Define solver box

Main Menu > Define > Models > Viscous

Laminar flow is the default. Therefore, we do not need to change anything in this menu.

Click Cancel.

Main Menu > Define > Models > Energy

For incompressible flow, the energy equation is decoupled from the continuity and momentum equations. We need to solve the energy equation only if we are interested in determining the temperature distribution. We will not deal with temperature in this example. So leave the Energy Equation unselected and click Cancel to exit the menu.

Define Material Properties

Main Menu > Define > Materials...

Change Density to 1.0 and Viscosity to 2e-3. We specified these values under Problem Specification. We will take both as constant.

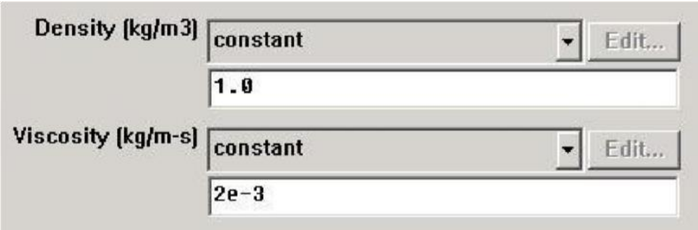


Figure 7. Material properties box

Click **Change/Create**.

Close the window.

Define Operating Conditions

Main Menu > Define > Operating Conditions...

- Note! For all flows, FLUENT uses gauge pressure internally. Any time an absolute pressure is needed, it is generated by adding the operating pressure to the gauge pressure. We will use the default value (1atm=101,325Pa=1bar) as the **"Operating Pressure"**.

Click Cancel to leave the default in place.

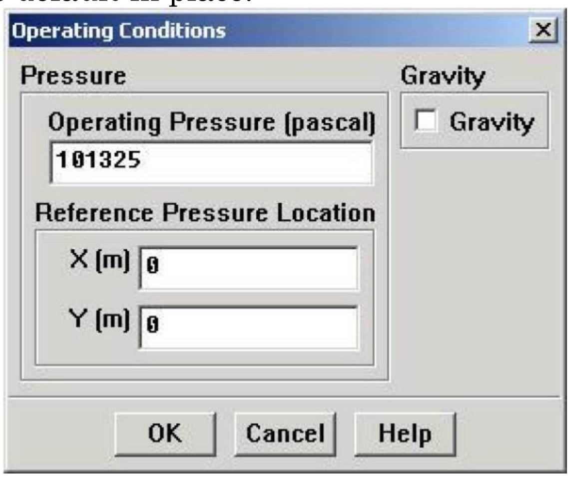


Figure 8. Operating conditions box

Define Boundary Conditions

We will now set the value of the velocity at the **inlet** and pressure at the **outlet**.

Main Menu > Define > Boundary Conditions...

- We note here that the four types of boundaries we defined are specified as zones on the left side of the *Boundary Conditions Window*. The *centerline* zone should be selected by default. Make sure it is, and then make sure the Type of this boundary is selected as axis and click set.... Notice that there is nothing to set for the axis. Click OK.
- Move down the list and select inlet under Zone. Note that FLUENT indicates that the Type of this boundary is velocity-inlet. Recall that the boundary type for the "inlet" was set in GAMBIT. If necessary, we can change the boundary type set previously in *GAMBIT* in this menu by selecting a different type from the list on the right.

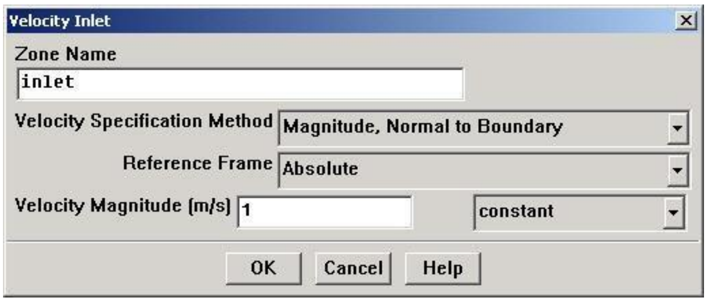


Figure 9. Inlet boundary condition box

Click on **Set**.... Enter 1 for Velocity Magnitude. Click **OK**. This sets the velocity of the fluid entering at the left boundary.

The **absolute** pressure at outlet is **1atm**. Since the operating pressure is set to **1atm**, the **outlet gauge pressure = outlet absolute pressure – operating pressure = 0.** Choose **outlet** under Zone. The type of this boundary is **"pressure-outlet"**. Click on **Set**.... The default value of the Gauge Pressure is "0". Click Cancel to leave the default in place.

Lastly, click on **wall** under Zones and make sure Type is set as **"wall"**. Click on each of the tabs and note that only momentum can be changed under the current conditions. This will not be so under later exercises so make a note of the location of these options. Click **OK**.

Click **Close** to close the *Boundary Conditions* menu.

Step 5: Solve!

We will use a second-order discretization scheme.

Solution

Main Menu > Solve > Controls > Solution...

Change Momentum to Second Order Upwind.

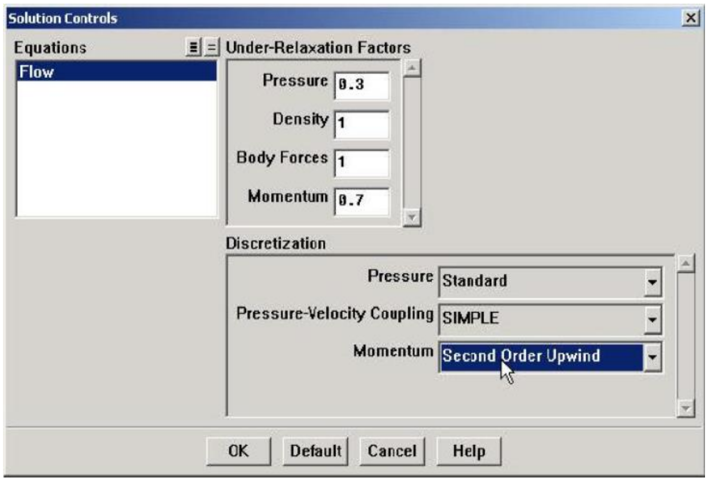


Figure 10. Solution control box

Click **OK**.

Initialization

Set Initial Guess

Initialize the flow field to the values at the inlet:

Main Menu > Solve > Initialize > Initialize...

In the *Solution Initialization* menu that comes up, choose inlet under Compute From.

The Axial Velocity for *all* cells will be set to 1 m/s, the Radial Velocity to 0 m/s and the Gauge Pressure to 0 Pa. These values have been taken from the inlet boundary condition.

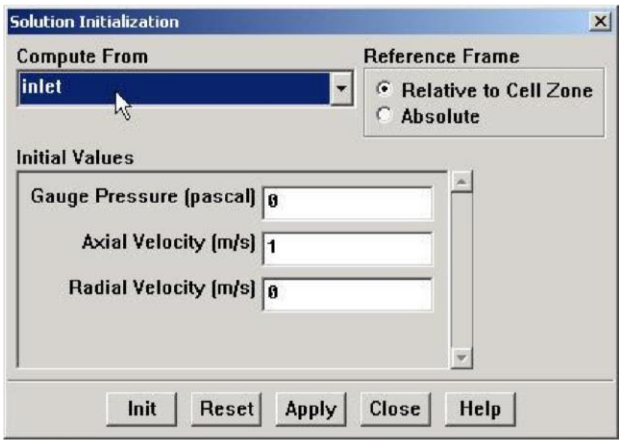


Figure 11. Initialization box

Click **Init**. This completes the initialization. Close the window.

Set Convergence Criteria

FLUENT reports a residual for each governing equation being solved. The residual is a measure of how well the current solution satisfies the discrete form of each governing equation. We will iterate the solution until the residual for each equation falls below 1e-6.

Main Menu > Solve > Monitors > Residual...

Change the residual under Convergence Criterion for continuity, x-velocity, and y-velocity, all to 1e-6.

Also, under Options, select Plot. This will plot the residuals in the graphics window as they are calculated.

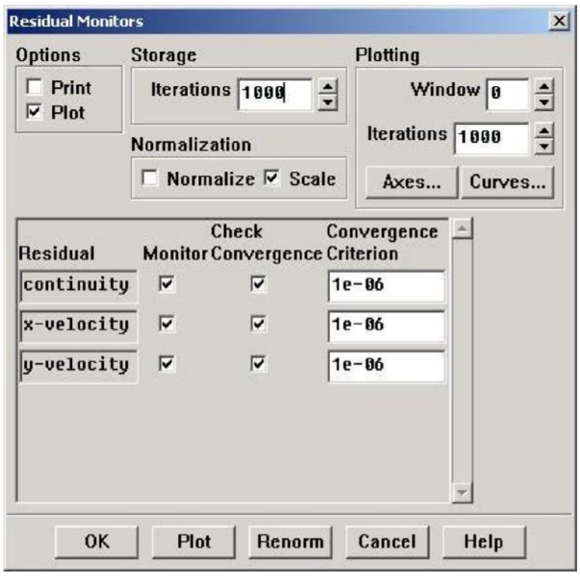


Figure 12. Residual monitors box

Click OK.

This completes the problem specification. Save your work:

Main Menu > File > Write > Case...

Type in *pipe.cas* for Case File. Click OK. Check that the file has been created in your working directory. If you exit FLUENT now, you can retrieve all your work at any time by reading in this case file.

Simulation

Iterate Until Convergence

Start the calculation by running 100 iterations:

Main Menu > Solve > Iterate...

In the *Iterate Window* that comes up, change the Number of Iterations to 100. Click Iterate.
The residuals for each iteration are printed out (on text command window), as well as plotted in the graphics window as they are calculated.

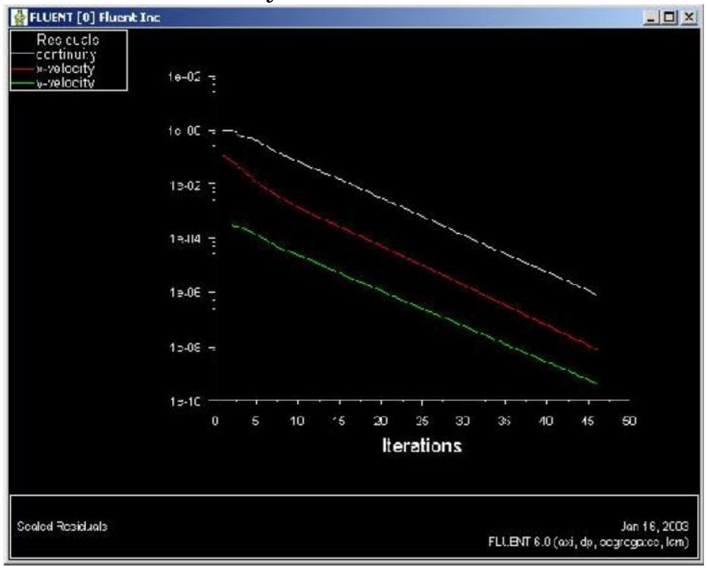


Figure 13. Residual profiles progress

The residuals fall below the specified convergence criterion of 1e-6 in about 46 iterations. Actual number of convergence steps may vary slightly.

```
  iter     time/iter
!   46 solution is converged
```

Figure 14. Convergence displayed message

Save the solution to a data file:

Main Menu > File > Write > Data...

Enter *pipe.dat* for Data File and click OK. Check that the file has been created in your working directory. You can retrieve the current solution from this data file at any time.

Step 6: Analyze Results

Centerline Velocity

We will 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.

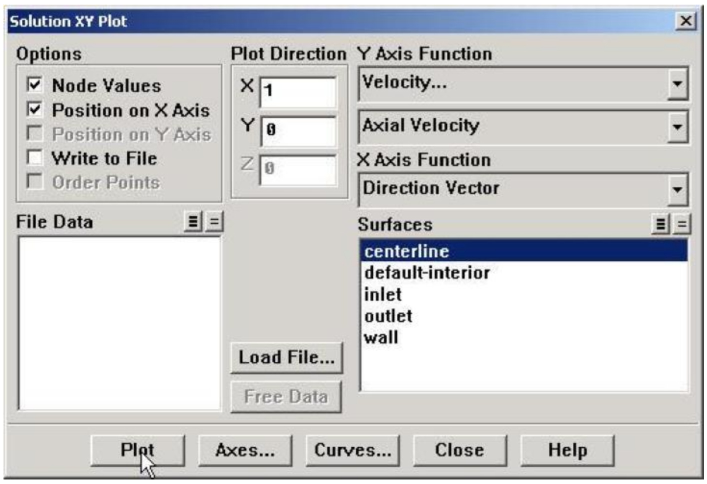


Figure 15. XY Plotting box

Click **Plot**.

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

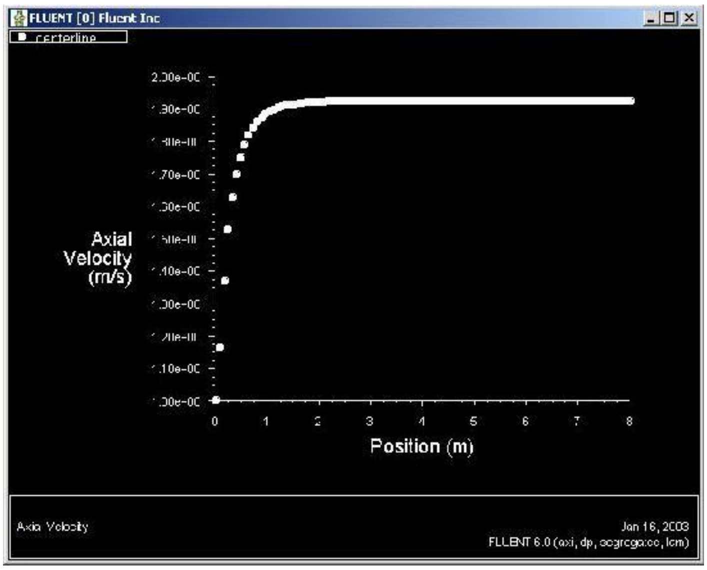


Figure 16. XY Plot : axial velocity profile

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 will 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**.

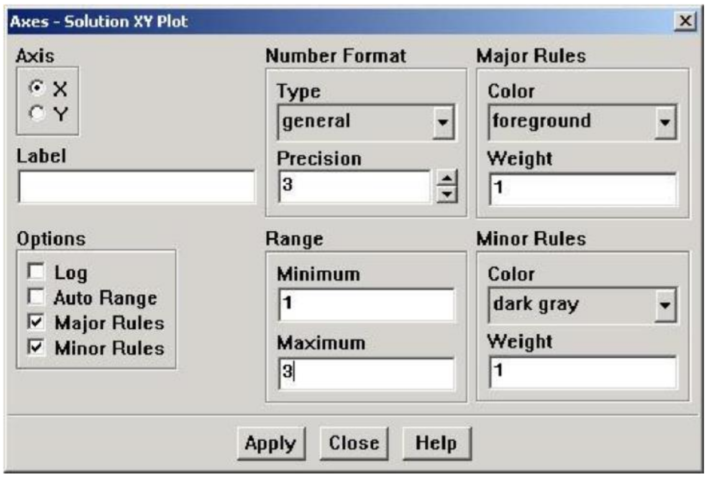


Figure 17. Change axes XY Plot displaying

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 Y 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*=3m and the centerline velocity in this region is *1.93 m/s*.

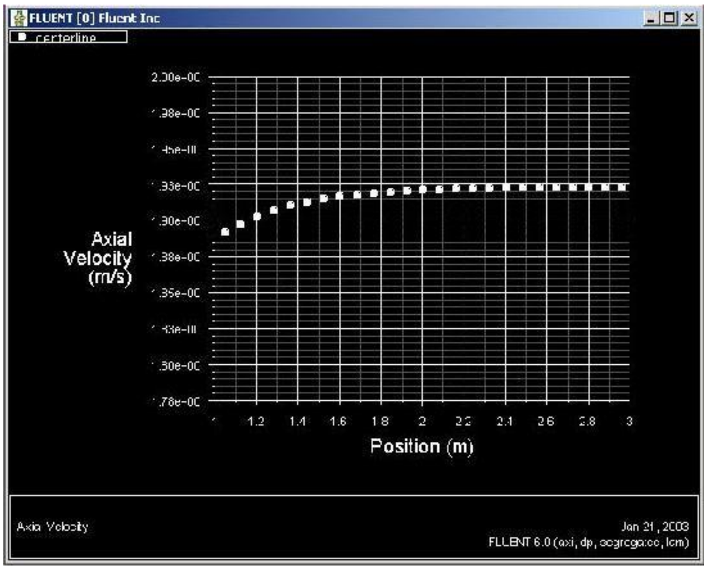


Figure 18. Changing display range of axes XY plot

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 is 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 965 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

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

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.

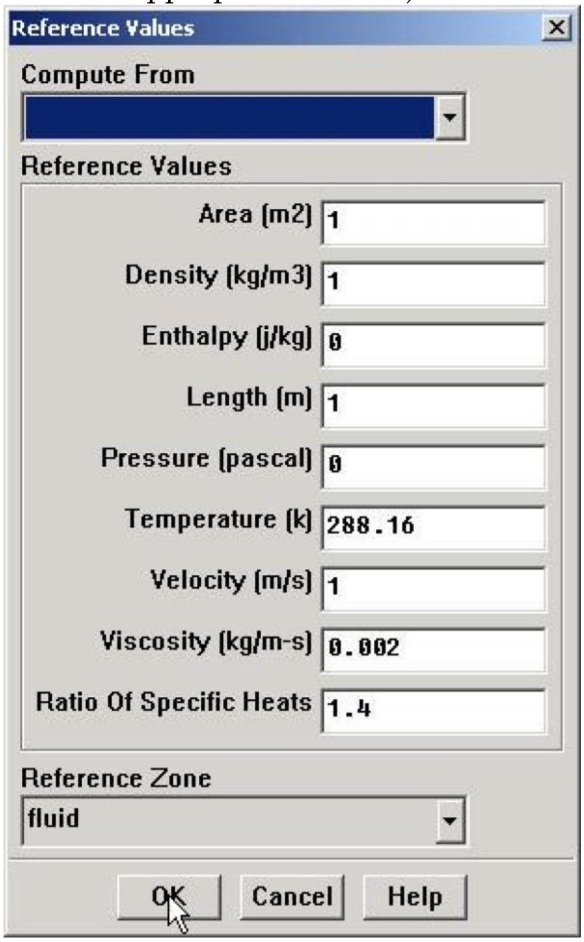


Figure 19. Reference values box

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 Options 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 Range box (remember to de-select Auto-Range first if it is checked).

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

Friction factor

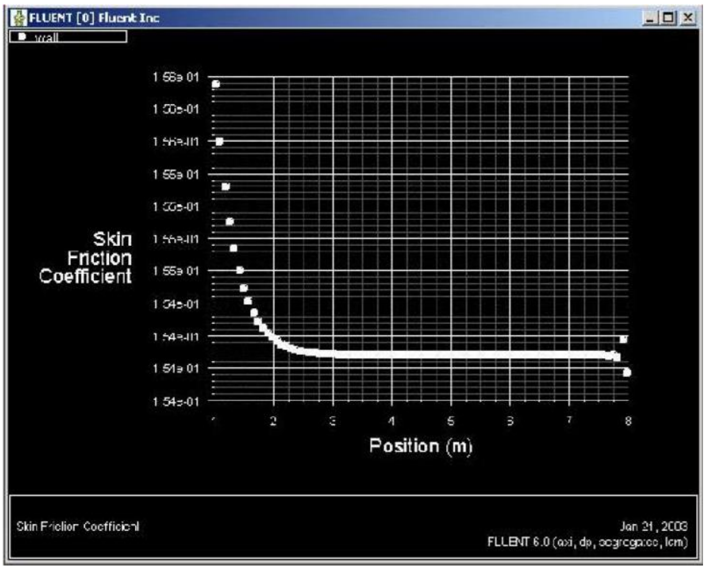


Figure 20. XY plot: friction factor profile

We can see that the fully developed region is reached at around x=3.0m 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 will 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)

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

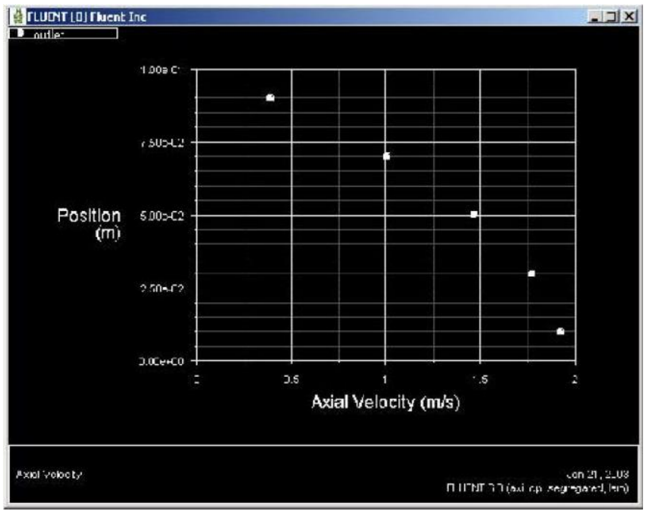


Figure 21. XY plot: Axial velocity profile

Does this look like a 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=0.6m (x/D=3) and x=0.12m (x/D=6) to the above plot. First, create a line at x=0.6m using the *Line/Rake* tool:

Main Menu > Surface > Line/Rake

We will create a straight line from (x0,y0)=(0.6,0) to (x1,y1)=(0.6,0.1). Select Line Tool under Options. Enter x0=0.6, y0=0, x1=0.6, y1=0.1. Enter line1 under New Surface Name. Click Create.

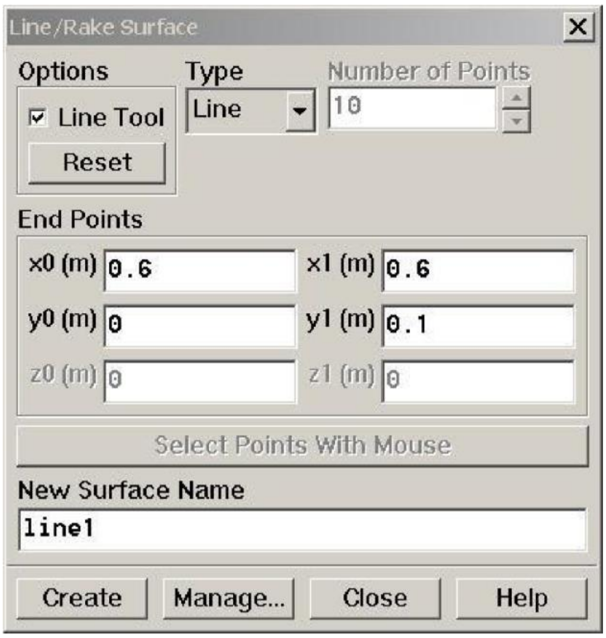


Figure 22. Creating new faces

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=0.6m. (Click here to review the zoom functionality discussion in step 4.) *line1* is the white vertical line to the right in the figure below.

Figure 23. New face created on x=0.6m

Similarly, create a vertical line called *line2* at x=1.2; (x0,y0)=(1.2,0) to (x1,y1)=(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=0.6m (x/D=3) and x=0.12m (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.

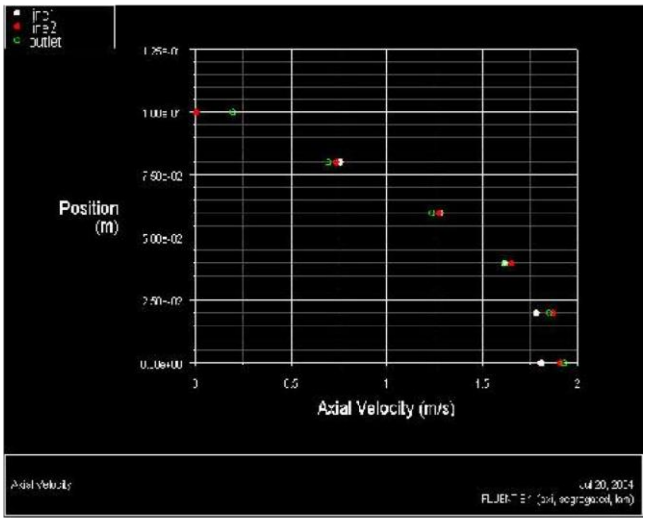


Figure 24. XY plot: overlaying axial velocity profiles

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 will see that this is not actually the case. The coarse grid used here does not 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.

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.

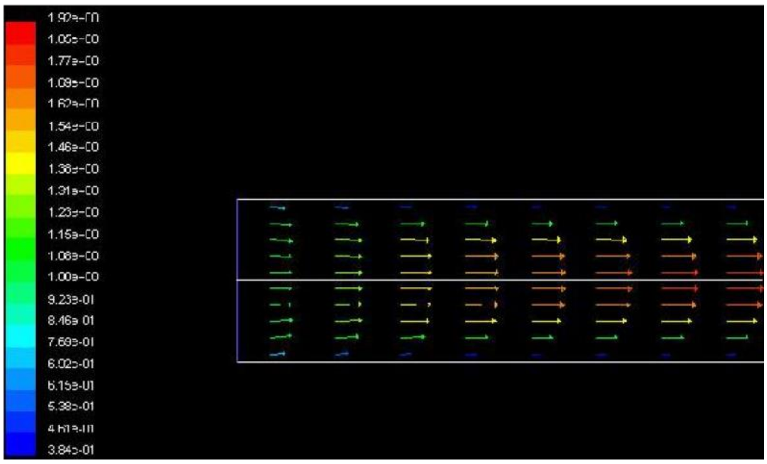


Figure 25. View of velocity vectors

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.

Step 7: Refine Mesh

It is very important to assess the dependence of your results on the mesh used by repeating the same calculation on different meshes and comparing the results. We will re-do the previous calculation on a 100x10 mesh and compare the results with the 100x5 mesh used previously. If you prefer to skip the GAMBIT steps for modifying the mesh, download the 100x21 mesh (by right-clicking on the link) and go directly to the FLUENT analysis discussed in TP1.

Modify Mesh in GAMBIT

The 100x5 mesh is saved as *pipe.dbs* in your working folder. Copy and paste the file in the same folder. Rename *Copy of pipe.dbs* to *pipe2.dbs*. We will work with *pipe2.dbs* in order to retain *pipe.dbs* as is. Launch *GAMBIT* and browse to where pipe2.dbs is saved. Notice that under Session ID, *pipe2* is now listed. Select this and click Run. Note in the main menu bar that *pipe2* is the ID of this job. Files created during this session will have that prefix.
We will delete the face mesh, modify the edge meshes for the vertical edges and remesh the face. To delete the original face mesh, choose

Operation Toolpad > Mesh Command Button > Face Command Button > Delete Face Meshes

In the *Delete Face Meshes Window* that comes up, *uncheck* the Remove unused lower mesh box. This tells *GAMBIT* to remove the face mesh only and keep the edge meshes associated with the face mesh. Since we will be changing the mesh on only two edges of the rectangle, there is no need to redo the meshes for all four edges.

Select the only face of the rectangle by shift-clicking on it and then click Apply.

Figure 26. Delete mesh of face

Modify Edge Meshes

To change the number of divisions on the vertical edges from 5 to 10, choose:

Operation Toolpad > Mesh Command Button > Edge Command Button > Mesh Edges

Select the two vertical edges by holding down the Shift button, clicking on each in turn, and then releasing the Shift button. Select Interval count from the box under Spacing that says Interval size. Change the number in the box next to the Interval count box from 5 to 10.. then to 21.

Make sure that the Remove old mesh box is checked under Options. This will make sure that the old edge meshes are erased before the new edge meshes are created.

Click Apply.

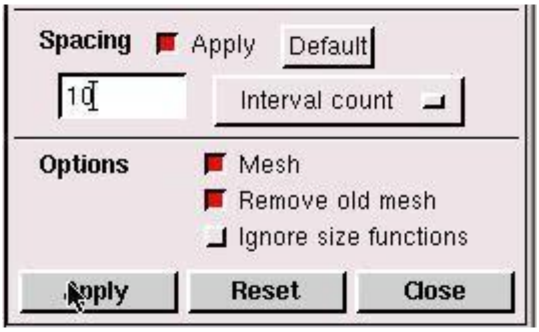


Figure 27. Change mesh spacing

Remember that you can zoom in by holding down Ctrl, dragging a box across the area you want to zoom in on, and then releasing Ctrl. Do this now and make sure that the vertical edges have 10 divisions, then 21 divisions.

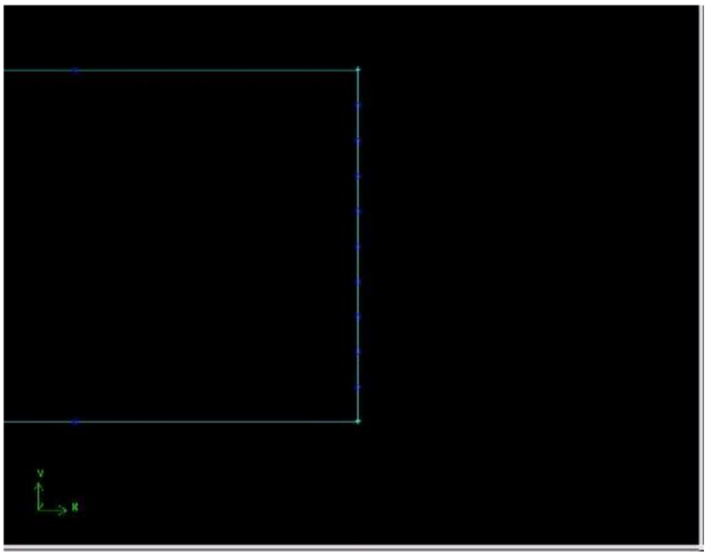


Figure 28. View of new meshing

Recreate Face Mesh

Operation Toolpad > Mesh Command Button > Face Command Button > Mesh Faces Shift-click on the face in the *Graphics Window* to select it. Click Apply.

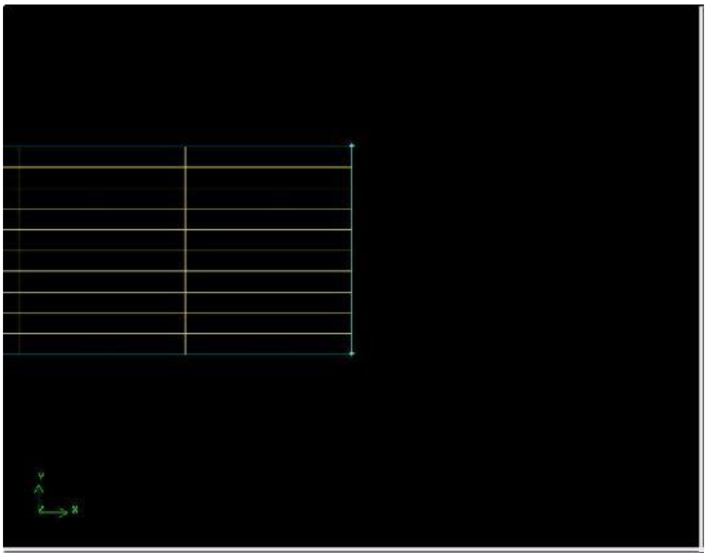


Figure 29. View of new mesh face

Save & Export
Main Menu > File > Save

Main Menu > File > Export > Mesh...

Type in pipe2.msh for the File Name:. Select Export 2d Mesh option. Click Accept.

Finer Mesh Analysis

Repeat steps 4 and 5 of this tutorial with the 100x10 mesh (a tad on the repetitious side but consider it good practice).

One you obtain the solution, plot the variation of the centerline velocity along the *x*-direction as described in step 6. Compare this result with that obtained on the previous mesh which is stored in the *vel.xy* file created earlier. To do this, after centerline velocity has been plotted, click on Load File... in the *Solution XY Plot* window.

Navigate to your working folder if necessary and click on *vel.xy* and OK. Click Plot.

In the graphics window, we can see both of the lines plotted in the same window.

Adjust the axes so that you can zoom in on the beginning of the fully developed region.

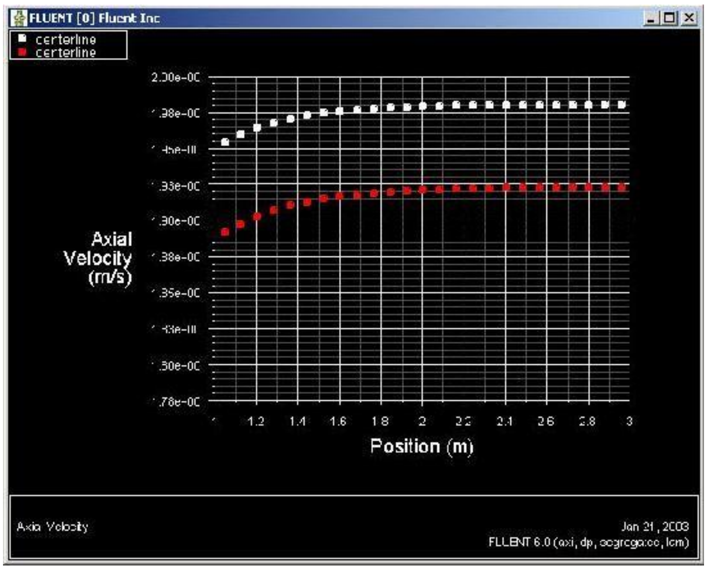


Figure 30. Compare axial velocity profiles of two meshes

In the centerline velocity plot above, the white and red symbols represent the results on the 100x10 mesh and 100x5 meshes, respectively. The centerline velocity in the fully developed region for the finer mesh is 1.98m/s. This value agrees better with the analytical value of 2m/s that the value of 1.93m/s obtained on the coarser mesh. Save the data for this plot as *vel2.xy*. The velocity result gets more accurate on refining the mesh as expected.

Plot the skin friction coefficient as described in step 6. Compare the result with that obtained on the 100x5 mesh by loading it from *cf.xy*.

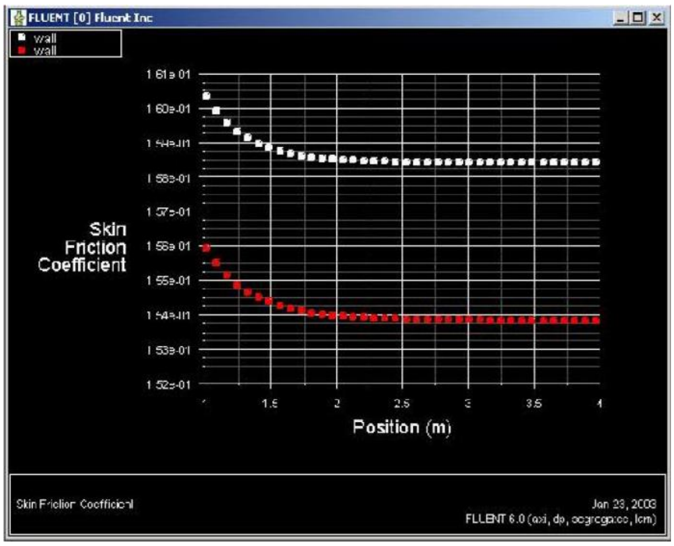


Figure 31. Compare friction factor profiles of two meshes

The finer mesh provides a skin friction coefficient of *0.159* in the fully-developed region, which is much closer to the theoretical value of *0.16* than the corresponding coarser mesh value of *0.154.* Save the data for this plot as *cf2.xy*.

Similarly, plot the velcoity profile at the outlet and compare with the coarser grid result in *out.xy*. The two results compare well with the greatest deviation occurring near the centerline. Save the data for this plot as *out2.xy*.

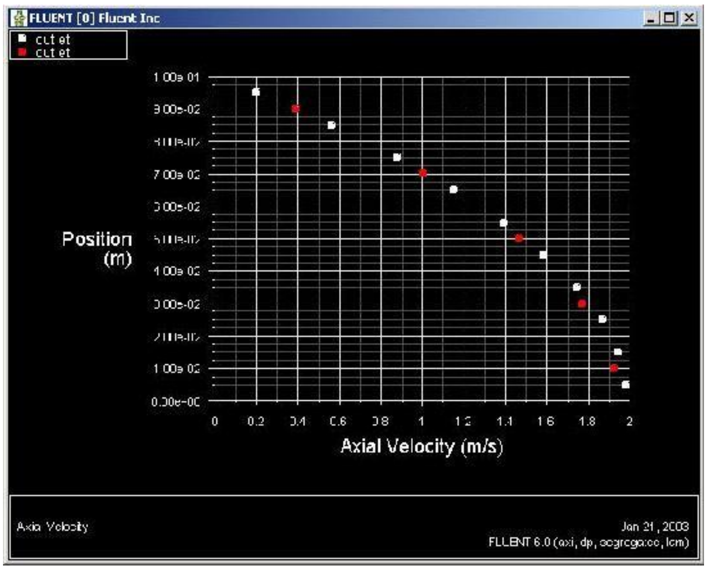


Figure 32. Compare the outlet velocity profiles of two meshes

If you repeat the calculation on a 100x21 mesh (as in TP1-5D presentation), you will see that the results on the two finest meshes are grid-independent to a high level of accuracy. In the plots below, the white, red and green symbols correspond to the 100x21, 100x10 and 100x5 meshes, respectively.

Velocity along centerline:

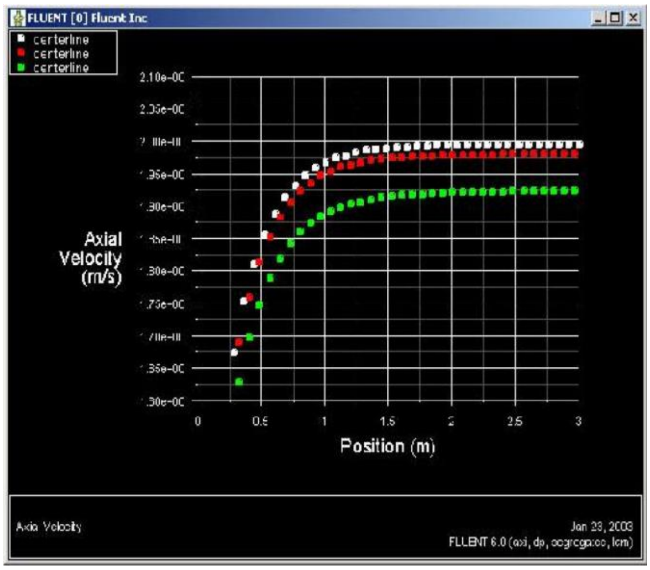


Figure 33. Compare three meshes : axial velocity profiles

Skin Coefficient :

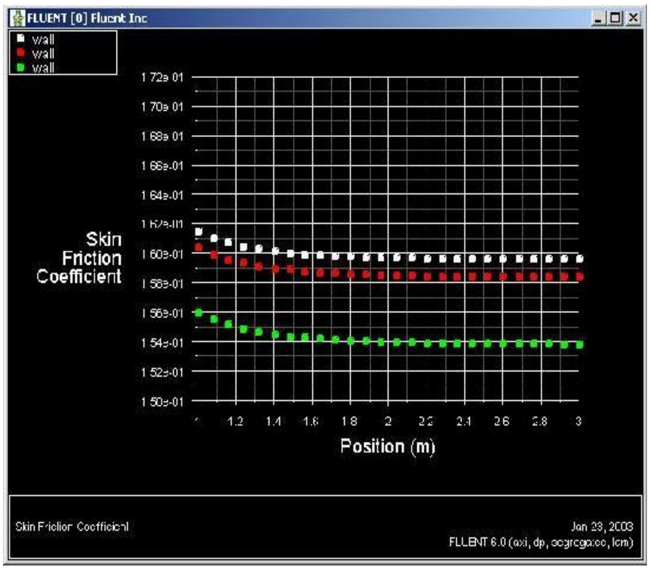


Figure 34. Compare three meshes: friction coefficient profiles

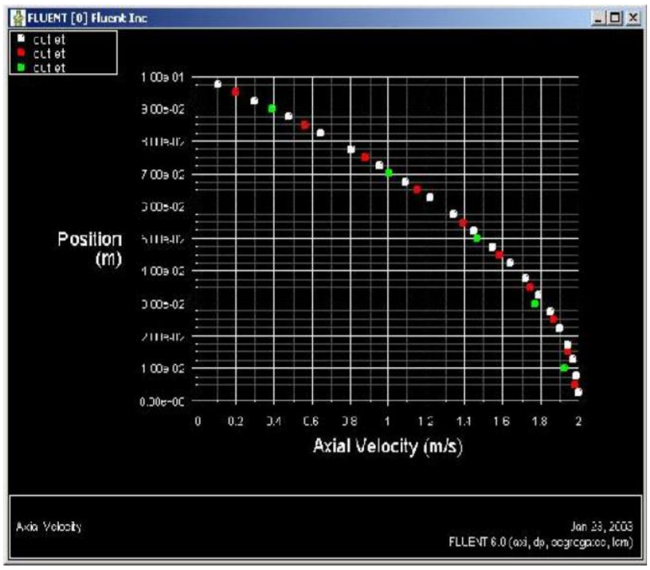


Figure 35. Compare three meshes : velocity profiles at outlet