Laminar Pipe Flow - Verification & Validation (FLUENT Post-processor)

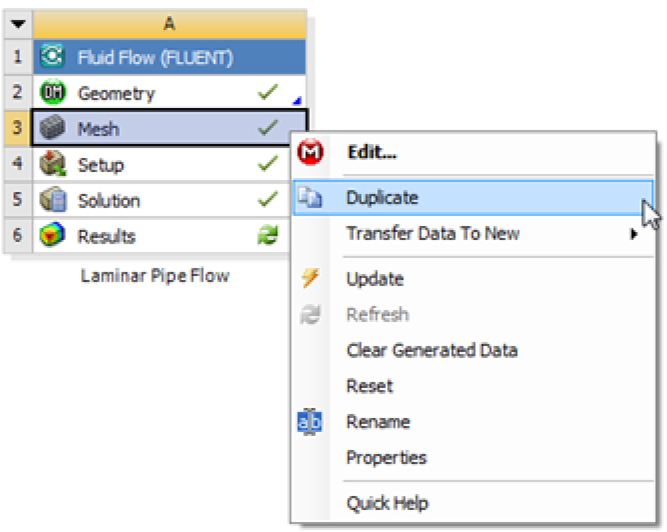
Author: Rajesh Bhaskaran, John Singleton, Cornell University

Step 7: Verification & Validation

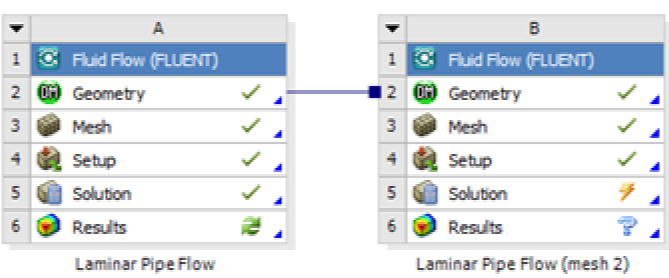
It is ***very important*** that you take the time to check the validity of your solution. This section leads you through some of the steps you can take to validate your solution.

Refine Mesh

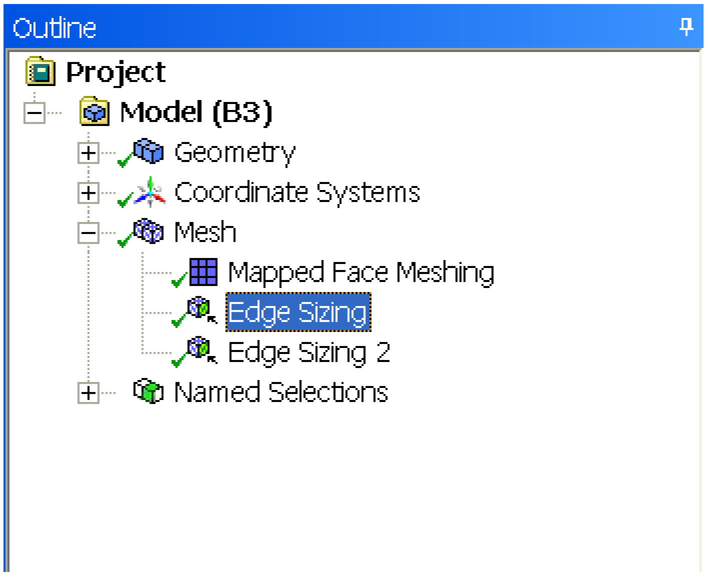
Let's repeat the solution on a finer mesh. For the finer mesh, we will increase the number of radial divisions from 5 to 10. In the ***Workbench Project Page*** right click on ***Mesh*** then click ***Duplicate*** as shown below.



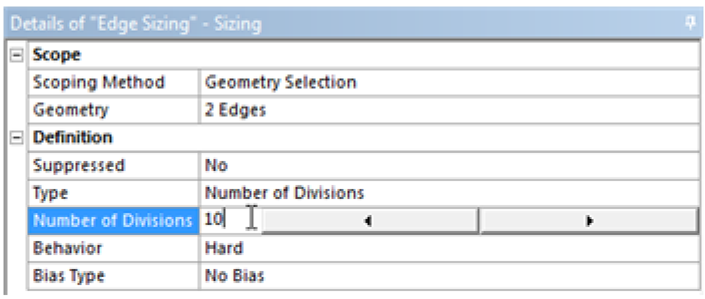
Higher Resolution Image
Rename the duplicate project to ***Laminar Pipe Flow (mesh 2)***. You should have the following two projects in your ***Workbench Project Page***.



Next, double click on the ***Mesh*** cell of the ***Laminar Pipe Flow (mesh 2)*** project. A new ANSYS Mesher window will open. Under ***Outline***, expand ***Mesh*** and click on ***Edge Sizing***, as shown below.

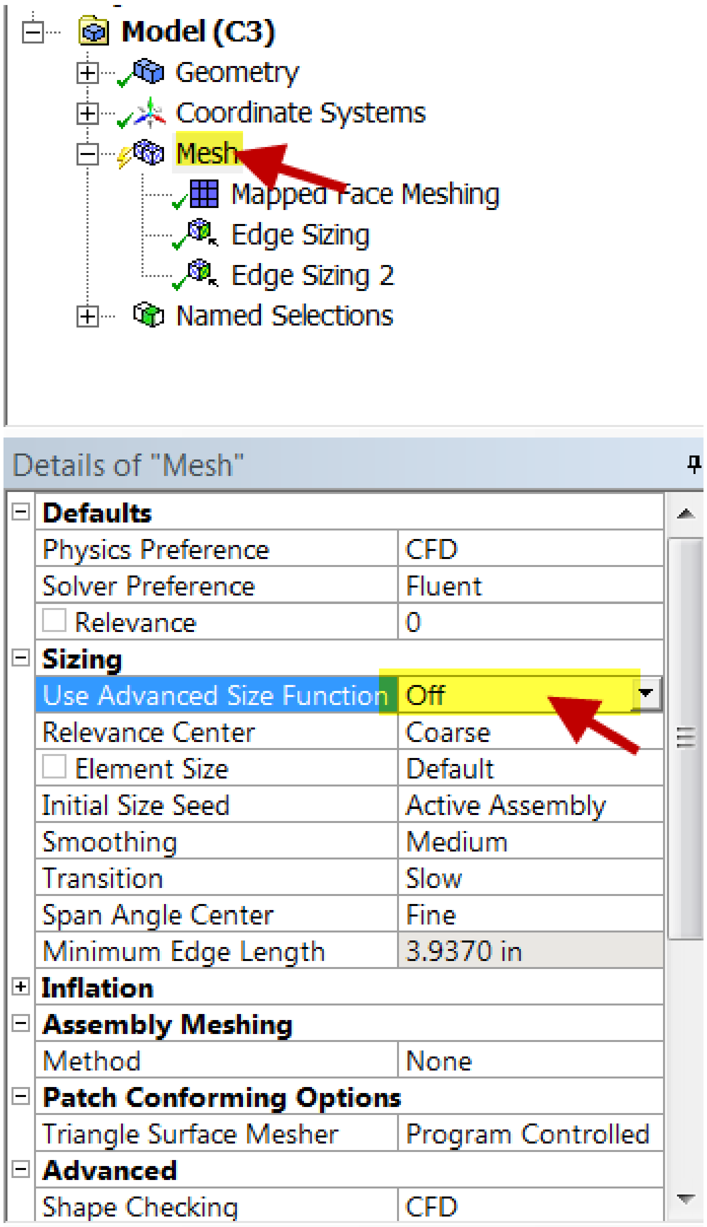


Under Details of "Edge Sizing", enter 10 for ***Number of Divisions***, as shown below.

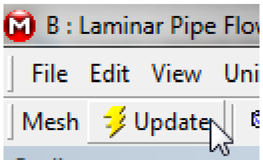


Higher Resolution Image

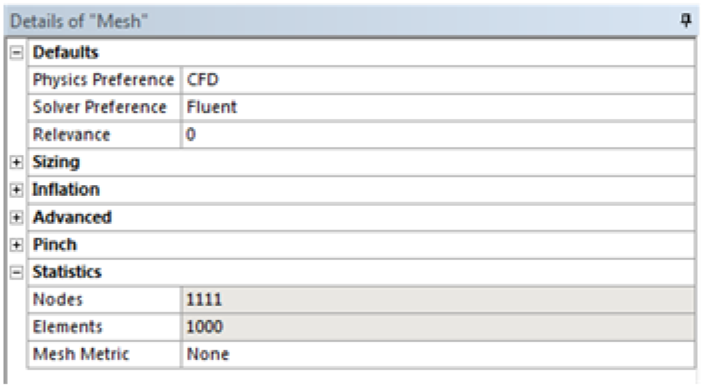
Sometimes, you need to turn-off "Advanced Size Function" under "Details of Mesh" to get the mesher to accept the modified settings. That way the Advanced Size Function feature will not over-ride your settings (this feature is useful for meshing complex geometries). Click ***Mesh*** in the tree and turn off Advanced Size Function under "Details of Mesh" as shown below.


Model (C3)
Geometry
Coordinate Systems
Mesh
Mapped Face Meshing
Edge Sizing
Edge Sizing 2
Named Selections
Details of "Mesh"
Defaults
Physics Preference
CFD
Solver Preference
Fluent
Relevance
0
Sizing
Use Advanced Size Function
Off
Relevance Center
Coarse
Element Size
Default
Initial Size Seed
Active Assembly
Smoothing
Medium
Transition
Slow
Span Angle Center
Fine
Minimum Edge Length
3.9370 in
Inflation
Assembly Meshing
Method
None
Patch Conforming Options
Triangle Surface Mesher
Program Controlled
Advanced
Shape Checking
CFD

Then, click ***Update*** to generate the new mesh.



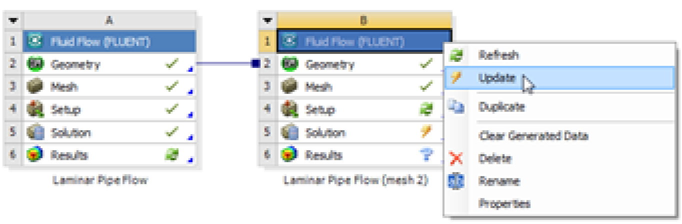
The mesh should now have 1000 elements (10 x 100). A quick glance of the mesh statistics reveals that there are indeed 1000 elements.



Higher Resolution Image

Compute the Solution

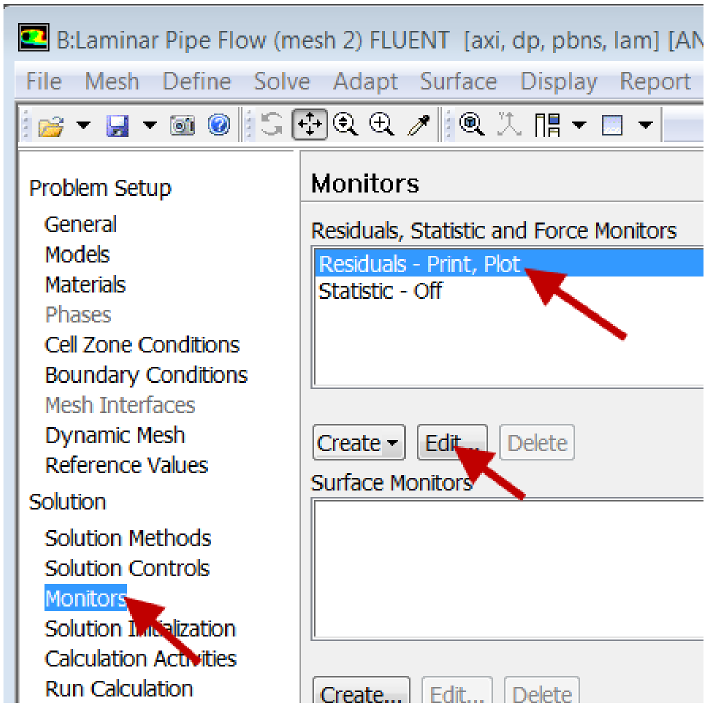
Close the ANSYS Mesher to go back to the ***Workbench Project Page***. Under ***Laminar Pipe Flow (mesh 2)***, right click on ***Fluid Flow (FLUENT)*** and click on ***Update***, as shown below.

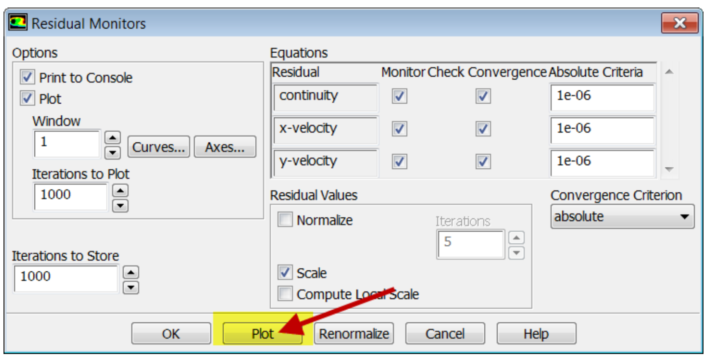


Higher Resolution Image

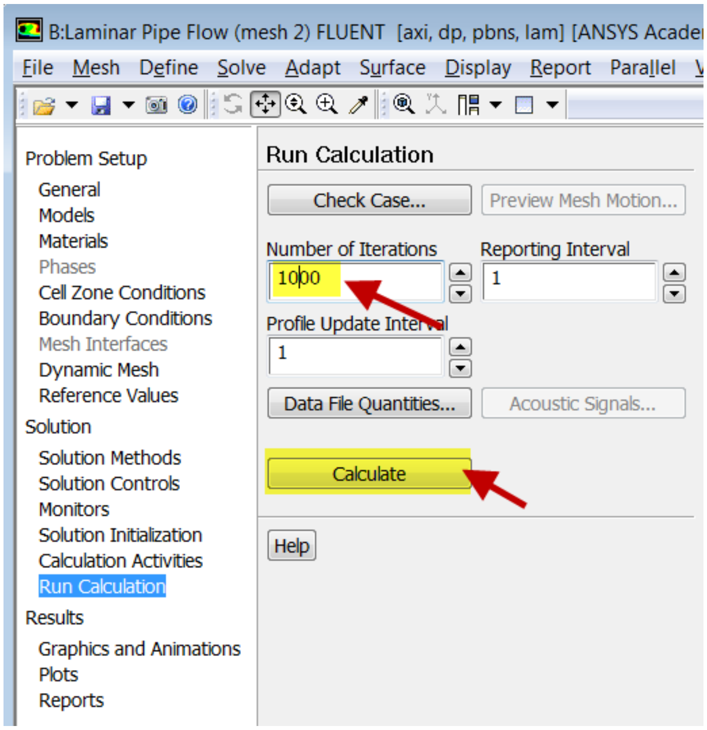
Now, wait a few minutes for FLUENT to obtain the solution for the refined mesh. After FLUENT obtains the solution, save your project.

It is necessary to check that the solution iterations have converged. Launch FLUENT by double clicking on ***Solution*** of the "Laminar Pipe Flow (mesh 2)" project in the ***Workbench Project Page***. After FLUENT launches, select ***Monitors > Residuals > Edit...*** and then ***Plot***, as shown in the images below.





It looks like my solution hasn't converged, so I need to run more iterations by selecting ***Run Calculation***. You may want to increase the number of iterations to, say, 1000. Ensure that you have a converged solution and save the project.

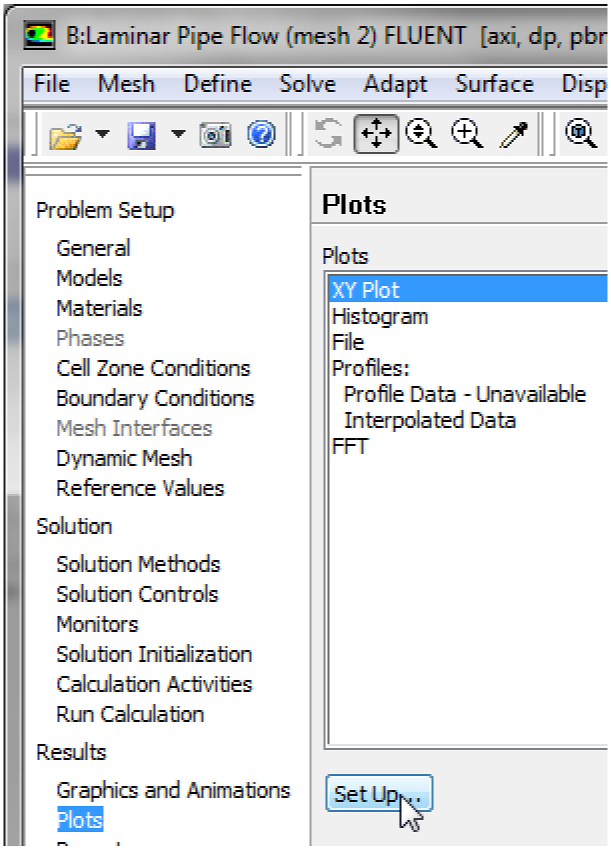


If you double-click on ***Results*** for mesh 2 in the project page, you'll see that all results have been updated for the new mesh. Also, you can drag ***Solution*** for the original mesh on to ***Results*** for mesh 2 in the project page. CFD-Post will automatically add the results from the original mesh to the plots for mesh 2. For instance, you will get the velocity profiles for both meshes in the same plot and you can export that to Excel and compare with the full-developed analytical solution.

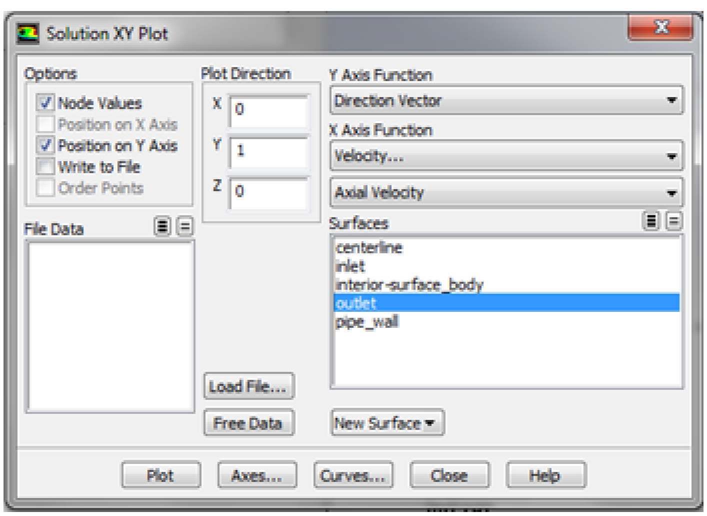
For instructions to compare results in the traditional FLUENT post-processor, click here and scroll down.

Velocity Profile

In order to launch FLUENT double click on the ***Solution*** of the "Laminar Pipe Flow (mesh 2)" project in the ***Workbench Project Page***. After, FLUENT launches ***(Click) Plots > XY Plot > SetUp...*** as shown in the image below.

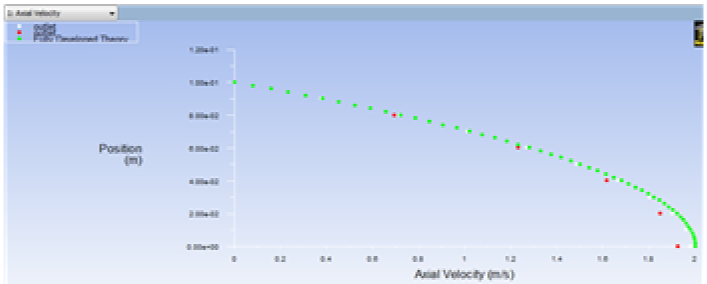


For this graph, the y axis of the graph will have to be set to the y axis of the pipe (radial direction). To plot the position variable on the y axis of the graph, uncheck ***Position on X Axis*** under ***Options*** and choose ***Position on Y Axis*** instead. To make the position variable the radial 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. Now, your ***Solution XY Plot*** menu should look exactly like the following image.



Higher Resolution Image

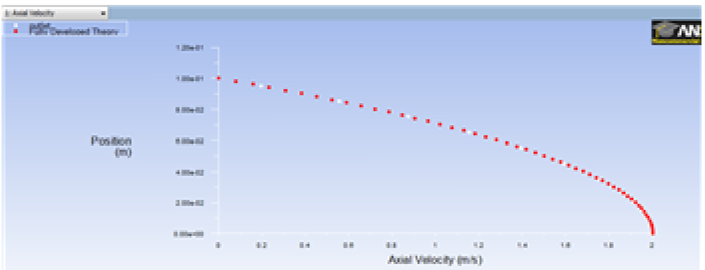
Since we would like to see how the results compare to the courser mesh and the theoretical solution, we will load the *profile.xy* file, which was created in the previous step. In order to do so, click ***Load File...*** in the ***Solution XY Plot*** menu, then select the *profile.xy* file. Click ***OK***, then click ***Plot*** in the ***Solution XY Plot*** menu. You should then obtain the following plot.



Higher Resolution Image
In the plot above the green dots correspond to the theoretical solution, the red dots correspond to the rough mesh (5 x 100), and the white dots correspond to the refined mesh (10 x 100). Note how the refined mesh results resemble the theory signicantly more than the rough mesh.

Further Verification

The plot below shows the results of a further refined mesh (20 radial x 100 axial) and the theoretical results.



Higher Resolution Image
Notice that for the further refined mesh, the results are almost indistinguishable from theory.

Go to Problem 1

See and rate the complete Learning Module

Go to all FLUENT Learning Modules