# Laminar Pipe Flow - Verification & Validation (FLUENT Postprocessor)

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
```

## 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.

Ŧ	A	
1	Fluid Flow (FLUENT)	
2	Geometry	× .
3	Mesh	× .
4	🍓 Setup	× .
5	Solution	× .
6	😥 Results	2
	Laminar Pipe Flow	

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.

Outline	<b>Ļ</b>
💼 Project	
🗄 🗟 Model (B3)	
🗄 √ 🏟 Geometry	
🗄 🏑 🙏 Coordinate Systems	
🚊 🗸 🚳 Mesh	
🚽 🛲 Mapped Face Meshing	
🖳 🗸 Edge Sizing	
🗸 🖧 Edge Sizing 2	
🗄 🗠 🎕 Named Selections	

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

D							
Ξ	Scope						
	Scoping Method	Geometry Selection	eometry Selection				
	Geometry	2 Edges	Edges				
Ξ	Definition						
	Suppressed	No					
	Туре	Number of Divisions					
	Number of Divisions	10 ] 🛛 🗍	<b>→</b>				
	Behavior	Hard					
	Bias Type	No Bias					

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
	🗄 🗝 🕸 Named Selections

D	etails of "Mesh"		Ą
Ξ	Defaults		
	Physics Preference	CFD	
	Solver Preference	Fluent	
	Relevance	0	
Ξ	Sizing		
	Use Advanced Size Function	Off 🚽 🔽	
	Relevance Center	Coarse	Ξ
	Element Size	Default	
	Initial Size Seed	Active Assembly	
	Smoothing	Medium	
	Transition	Slow	
	Span Angle Center	Fine	
	Minimum Edge Length	3.9370 in	
÷	Inflation		
Ξ	Assembly Meshing		
	Method	None	
Ξ	Patch Conforming Options	5	
	Triangle Surface Mesher	Program Controlled	
Ξ	Advanced		
	Shape Checking	CFD	Ŧ

Then, click Update to generate the new mesh.

🔯 B : Laminar Pipe Flow						
File	Edit	View	Uni			
Mesh	۶Ų	Jpdate				

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

De	etails of "Mesh"	ф.			
=	Defaults				
	Physics Preference	CFD			
	Solver Preference	Fluent			
	Relevance	0			
÷	Sizing				
Đ	Inflation				
Ŧ	Advanced				
Ŧ	Pinch				
Ξ	Statistics				
	Nodes	1111			
	Elements	1000			
	Mesh Metric	None			

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.

•		A		
1	8	Fluid Flow (FLUENT)		
2	69	Geometry	~	7
3	0	Mesh	1	
4	di,	Setup	~	1
5	-	Solution	~	
6		Results	æ	
		Laminar Pipe Flow		

#### 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.

File       Mesh       Define       Solve       Adapt       Surface       Display       Report         Image: Imag	B:Laminar Pipe Flow (m	esh 2) FLUENT [axi, dp, pbns, lam] [AN
Image:	File Mesh Define Solv	e Adapt Surface Display Report
Problem Setup       Monitors         General       Residuals, Statistic and Force Monitors         Models       Residuals, Statistic and Force Monitors         Materials       Phases         Cell Zone Conditions       Residuals - Print, Plot         Boundary Conditions       Statistic - Off         Mesh Interfaces       Dynamic Mesh         Dynamic Mesh       Create           Reference Values       Surface Monitors         Solution       Solution Controls         Monitors       Solution Intelization         Calculation Activities       Create         Run Calculation       Create	😂 🔻 🛃 🕶 🚳 🖉 🕄 (	Đ€€ ∥ ∥® 밨 ⊪ - □ -
Calculation Activities Run Calculation Create Edit Delete	Problem Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors Solution Interfaction	Monitors Residuals, Statistic and Force Monitors Residuals - Print, Plot Statistic - Off Create  Edit Delete Surface Monitors
	Run Calculation	Create Edit Delete

Residual Monitors					×
Options	Equations				
Print to Console	Residual	Monitor Ch	neck Convergen	ce Absolute Criteria	^
Plot	continuity	<b>v</b>		1e-06	
Window	x-velocity	<b>V</b>		1e-06	
Iterations to Plot	y-velocity	✓		1e-06	-
1000	Residual Values			Convergence Crit	erion
	Normalize	Ι	terations	absolute	•
Iterations to Store			5		
1000	Scale				
	Compute Log	al Scale			
ОК Р	ot Renorma	lize Ca	incel He	elp	

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.

💶 B:Laminar Pipe Flow (m	esh 2) FLUENT [axi, dp, pbns, lam] [ANSYS Acade
<u>File Mesh Define Solv</u>	e <u>A</u> dapt S <u>u</u> rface <u>D</u> isplay <u>R</u> eport Para <u>l</u> lel <u>V</u>
Eile       Mesh       Define       Solv         Image: Im	e <u>A</u> dapt <u>Su</u> rface <u>Display Report Parallel</u> <u>Y</u>
Reports	

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.

Solution XY Plot			×
Options	Plot Direction	Y Axis Function	
Vode Values	X O	Direction Vector	•
Position on X Axis		X Axis Function	_
Write to File	1	Velocity	•
Order Points	Z O	Axial Velocity	•
File Data 🔳 🚍		Surfaces	
		centerline inlet interior-surface_body	
		outet	
	Load File		
·	Free Data	New Surface *	
Plot	Axes	Curves Close Help	

#### 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