Sudden Expansion - Verification & Validation

Authors: Yong Wang & Said Elghobashi, UC Irvine Problem Specification 1. Pre-Analysis & Start-Up 2. Geometry 3. Mesh

- Physics Setup
 Numerical Solution
- 6. Numerical Results
- 7. Verification & Validation
- Exercises
- Comments

Verification and Validation

It is *very important* that you take the time to check the validity of your solution. Based on the velocity profile along the centerline, it can be seen that the max axis velocity in the small pipe is about 5.25e-1 m/s. However, in a full-developed laminar pipe flow, the analytical max axis velocity should be two times of the uniform inlet velocity, that is 5.54e-1 m/s. In other words, the results with the previous mesh are not good enough.

Refine Mesh

Let's repeat the solution on a finer mesh. For the finer mesh, the numbers of radial divisions will be twice as many as those used before. In the *Workbench Project Page* right click on *Mesh* then click *Duplicate* as shown below.



Rename the duplicate project to suddenExpansion (mesh2). You should have the following two projects in your Workbench Project Page.

•	А	
1	S Fluid Flow (Fluent)	
2	Geometry	× .
3	🍘 Mesh	× .
4	🎡 Setup	× .
5	Solution	× .
6	🥩 Results	2 🖌
	suddenExpansion	

Next, double click on the *Mesh* cell of the *suddenExpansion (mesh2)* project. A new ANSYS Mesher window will open. Under *Outline*, expand *Mesh* and click on *Edge Sizing*, as shown below.



Here we will increase the numbers of cells in the radial direction. You may have different names of sizings. In my case, the "Edge Sizing" corresponds to the AB and EF edges, and "Edge Sizing 4" corresponds to the CD edge. Double the *Number of Divisions* of "Edge Sizing"

Details of "Edge Sizing" - Sizing 7						
	Scope					
	Scoping Method	Geometry Selection				
Geometry 2 Edges						
	Definition					
	Suppressed	No				
	Туре	Number of Divisions				
	Number of Divisions	40	•	→		
	Behavior	Hard				
	Bias Type	No Bias	;			

and "Edge Sizing 4"

Details of "Edge Sizing 4" - Sizing 4						
Ξ	Scope					
	Scoping Method	Geometry Selection				
Geometry		1 Edge				
Ξ	- Definition					
Suppressed N Type N		No				
		Number of Divisions				
	Number of Divisions	80	•			
	Behavior	Hard				
	Bias Type No Bias					

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.



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

Mesh 🔰 Update

The mesh should now have 44000 elements. A quick glance of the mesh statistics reveals that there are indeed 44000 elements.

Details of "Mesh"						
Ŧ	Defaults					
Ŧ	Sizing					
Ŧ	Inflation					
Ŧ	Assembly Meshing					
Ŧ	Patch Conforming Options					
Ŧ	Advanced					
Ŧ	Defeaturing					
Ξ	Statistics					
	Nodes	44681				
	Elements	44000				
	Mesh Metric	None				

Compute the Solution

Close the ANSYS Mesher to go back to the *Workbench Project Page*. Under *suddenExpansion (mesh2)*, right click on *Fluid Flow (FLUENT)* and click on *Update*, as shown below.



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 "*suddenExpansion (mesh2)*" project in the *Workbench Project Page*. After FLUENT launches, select *Monitors > Residuals > Edit...* and then *Plot*, as shown in the images below.



Residual Monitors							
Options	Equations						
Print to Console	Residual Monitor Check Convergence Ab			ce Absolute Criteria	*		
V Plot	continuity	V	\checkmark	1e-06			
Window	x-velocity		V	1e-06			
Iterations to Plot	y-velocity	V	\checkmark	1e-06	-		
	Residual Values			Convergence Crit	terion		
	Normalize	1	terations	absolute	•		
Iterations to Store			5				
1000	Scale						
	Compute Lor	scale					
OK Plot Renormalize Cancel Help							

If the solution hasn't converged, one needs to run more iterations by selecting *Run Calculation*. You may want to increase the number of iterations. Ensure that you have a converged solution and save the project. Then, if you double-click on *Results* for mesh2 in the project page, you'll see that all results have been updated for the new mesh.

Velocity profile

The plot below shows the profile of the axial velocity along the centerline (y = 0 m) with the refined mesh. It can be seen that the max velocity in the small pipe is about 5.50e-1 m/s, which is close to the analytical solution (5.54e-1 m/s) and better than that of the previous mesh.



Here we will create a straight line for the expansion entrance, which is from (x0, y0) = (0, 0) to (x1, y1) = (0, 1). Select *Line* under *Surface*. Enter x0 = 0, y0 = 0, x1 = 0, y1 = 1. Enter expansion under *New Surface Name*. Click *Create*.

Line/Rake Surface						
Options Type Number of Points Line Tool Reset						
End Points						
x0 (m) 0 x1 (m) 0						
y0 (m) 0 y1 (m) 1						
z0 (m) 0 z1 (m) 0						
Select Points with Mouse						
New Surface Name						
expansion						
Create Manage Close Help						

Then, plot the velocity profile along the *expansion* line. In the *Solution XY Plot* menu make sure that *Position on X Axis* is selected, and *X* is set to 0 and *Y* is set to 1.

Solution XY Plot					
Options V Node Values Position on X Axis Position on Y Axis Write to File Order Points File Data	Plot Direction X 0 Y 1 Z 0	Y Axis Function Velocity Axial Velocity X Axis Function Direction Vector Surfaces Centerline expansion inlet interior-surface_body line-0 outlet wall New Surface			
Plot Axes Curves Close Help					

Then, click *Plot* and you should obtain the following output.



You can also plot axial velocity profiles at x = 1 m, 2 m, 3 m, 4 m, 5 m and 6 m as shown below. The numbers of the lines also indicate the distances from the expansion entrance to these lines.



As you can see from the velocity profiles, the end of the recirculation region is at x = 5 m. In other words, the recirculation length for Re = 55.4 is about 5 times of the inlet radius, which agree well with the experimental data (*Hammad et al., Experiments in Fluids, 1999, 26:266-271*) shown below.



Streamline

First, click on *Graphics & Animations*. Next, double click on *Contours* which is located under *Graphics*. Select *Velocity* and *Stream Function* under *Contours of*, and change the number of *Levels* to 50, as shown below.

Contours	a term	23			
Options	Contours of				
Filled	Velocity	-			
✓ Node Values	Stream Function	•			
Auto Range	Min (kg/s) Max (kg/s)				
Clip to Range	0 0.1512765				
Draw Profiles	Surfaces				
	centerline				
Levels Setup	inlet				
50 1	line-0				
	wall				
Surface Name Pattern	New Surface				
Match	Surface Types				
	axis	*			
	exhaust-fan				
	fan	•			
Display Compute Close Help					

Then, click on *Display*. Zoom into the region near the expansion. The streamlines calculated with FLUENT are presented blow, and compared with the streamlines obtained via PIV experiment (*Hammad et al., Experiments in Fluids, 1999, 26:266-271*). It can be seen that the numerical result agree with the experiment one.



For more strict comparison, one should export the results from FLUENT, re-plot it in TECPLOT or some other post-processing software, and compare the value of stream function, redevelopment length and recirculation length carefully. One can also increase the inlet velocity so that Re = 200, and then compare the results with experiment data presented in the reference.

Go to Exercises

Go to all FLUENT Learning Modules