# **Sudden Expansion - Mesh**

Authors: Yong Wang & Said Elghobashi, UC Irvine

Problem Specification 1. Pre-Analysis & Start-Up 2. Geometry 3. Mesh 4. Physics Setup 5. Numerical Solution 6. Numerical Results 7. Verification & Validation Exercises Comments

# Mesh

In this section the geometry will be meshed. The cell numbers on the edges of the desired mesh are shown here:



In this section the geometry will be meshed. The cell numbers on the edges of the desired mesh are shown here:

#### Launch Mesher

In order to begin the meshing process, go to the Workbench Project Page, then (Double Click) Mesh.

You may want to generate a preliminary mesh by right clicking on Mesh (upper window) and then "Generate Mesh".

#### **Mapped Face Meshing**

Here we are interested in creating a grid style of mesh that can be mapped to a rectangular domain. This meshing style is called *Mapped Face Meshing*. In order to incorporate this meshing style (*Click*) *Mesh Control* > *Mapped Face Meshing* as can be seen below.

😭 A : Fluid Flow (FLUENT) - Meshi	ng [ANSYS Academic Teaching Advan
File Edit View Units Tools	Help 🗍 🏥 📷 🛆 🥑 🕶 🕼 V
Mesh 🖸 Update 🛛 🍘 Mesh 🔻	🔍 Mesh Control ▼ 🔤 ,II Metric Gra
Outline	The matched the ma
Project Model (A3)  Model (A3)  Coordinate Systems  Mesh	<ul> <li>Sizing</li> <li>Contact Sizing</li> <li>Refinement</li> <li>Mapped Face Meshing</li> <li>Match Control</li> </ul>
	Pinch A Inflation
	Gap Tool

Now, the *Mapped Face Meshing* still must be applied to the pipe geometry. In order to do so, first click on the pipe body which should then highlight green. Next, (*Click*) *Apply* in the *Details of Mapped Face Meshing* table, as shown below.

D	etails of "Mapped Fac	e Meshing" - Mapped Face Me	eshing 🛛	
Ξ	Scope			
	Scoping Method	Geometry Selection		
	Geometry	Apply	Cancel	
=	Definition			
	Suppressed	No		
	Method	Quadrilaterals		
	Constrain Boundary	No		
-	Advanced			
	Specified Sides	None		
	Specified Corners	None		
	Specified Ends	None		

After clicking Apply, you should see "Geometry: 1Face". Now, (Click) Mesh > Generate Mesh as can be seen below, and generate a rough mesh.



### **Edge Sizing**

The desired mesh has specific number of divisions along the radial and the axial direction. In order to obtain the specified number of divisions *Edge Sizing* must be used. The divisions along the axial direction will be specified first. Now, an *Edge Sizing* needs to be inserted. First, *(Click) Mesh Control > Sizing* as shown below.



Now, the geometry and the number of divisions need to be specified. First (*Click*) *Edge Selection Filter*, , in the upper toolbar. Hold down the "Ctrl" button and then click the edges **AF** and **BG**. Both sides should highlight green. Next, hit **Apply** under the **Details** of **Sizing** table as shown below.

De	etails of "Sizing" - Sizing		ą
Ξ	Scope		
	Scoping Method	Geometry Selection	
	Geometry	Apply	Cancel
Ξ	Definition		
	Suppressed	No	
	Туре	Element Size	
	Element Size	Default	
	Behavior	Soft	
	Curvature Normal Angle	Default	
	Growth Rate	Default	

Now, change Type to Number of Divisions as shown in the image below.

Details of "Edge Sizing" - Sizing 📮		
😑 Scope		
	Scoping Method	Geometry Selection
	Geometry	2 Edges
Ξ	Definition	
	Suppressed	No
	Туре	Element Size 💌
	Element Size	Element Size
	Behavior	Number of Divisions
	Curvature Normal Angle	Default
	Growth Rate	Default
	Bias Type	No Bias

Then, set *Number of Divisions* to 100 as shown below.

D	Details of "Edge Sizing" - Sizing 🛛 🐥			
Ξ	Scope			
	Scoping Method	Geome	try Selection	
	Geometry	2 Edge	\$	
Ξ	Definition			
	Suppressed	No		
	Туре	Numbe	r of Divisions	
	Number of Divisions	100	•	→
	Behavior	Soft		
	Curvature Normal Angle	Default	t	
	Growth Rate Default			
Bias Type No Bias				

Change Behavior to Hard for both Edge Sizing's.

De	Details of "Edge Sizing" - Sizing 4		
Ξ	Scope		
	Scoping Method	Geometry Selection	
	Geometry	2 Edges	
Ξ	Definition		
	Suppressed	No	
	Туре	Number of Divisions	
	Number of Divisions	100	
	Behavior	Soft 🔹	
	Curvature Normal Angle	Soft	
	Growth Rate	Deraut	
	Bias Type	No Bias	

Follow the same procedure as for the edge sizing on the other edges. Make sure to (*Click*) *Mesh Control* > *Sizing* every time. Then, generate the mesh by clicking *Mesh* > *Generate Mesh*. You should obtain the following mesh.



Detail around the expansion entrance FG is presented blow.



Mesh statistics can be found by clicking on *Mesh* in the tree and then by expanding *Statistics* under the *Details of Mesh* table. It can be seen that there are 22000 elements in the mesh above.

#### **Create Named Selections**

Here, the edges of the geometry will be given names so one can assign boundary conditions in Fluent in later steps. The left side of the pipe will be called "Inlet" and the right side will be called "Outlet". The bottom side will be called "CenterLine" and the other edges are called "Wall", as shown in the image below.

	Wall	
	1	
Inlet		Outlet
	CenterLine	

In order to create a named selections first Click *Edge Selection Filter*, . Then click on the left side of the rectangle and it should highlight green. Next, right click the left side of the rectangle and choose *Create Named Selection* as shown below.

	Insert
	Go To
•	Isometric View
ISO	Set
ISO	Restore Default
Q	Zoom To Fit
	Cursor Mode
	View
1Q	Look At
Ť	Select All
0	Suppress Body
P	Hide Body
*	Create Coordinate System
÷¢	Create Named Selection
<u>-</u>	Create Named Selection Refresh Geometry

Select the left edge and right click and select Create Named Selection. Enter Inlet and click OK, as shown below.

Selection Name				
Inlet				
Apply selected geometry				
O Apply geometry items of same:				
Size				
П Туре				
Location X				
Location Y				
Location Z				
Apply To Corresponding Mesh Nodes				
OK Cancel				

Now, create named selections for the remaining edges and name them according to the diagram. Since the "Centerline" consists of two edges thus when naming the centerline you should use "*Ctrl*" to select both parts. For the "Wall", use "*Ctrl*" to select three edges.

#### Save, Exit & Update

First save the project in the Mesher window. Next, close the Mesher window. Then, go to the Workbench Project Page and click the Update Project butto

## n, 🦩 Update Project

If the Progress window shows error message like this: "Model information is incompatible with incoming mesh", right click on Setup in the workbench and select "Reset", then Update Project again.

#### Go to Step 4: Physics Setup

Go to all FLUENT Learning Modules