Problem Statement

Consider a fluid flow through a sudden expansion in an axisymmetric pipe. The flow is laminar and axisymmetric. Due to symmetry, the computational domain covers only half of the pipe. BD is the axis of symmetry. The radius R1 = 1m and R2/R1 = 2.

L1/R1 = 20 and L2/R2 = 50. The inlet velocity at AB is uniform, U1 = 0.277 m/s. The fluid exhausts into the ambient atmosphere which is at a pressure of 1 atm at CD. The density ρ = 1kg/m3, and the dynamic viscosity is: μ = 0.01 kg/(ms).

The Reynolds number Re at AB = (2 R1 ρ U1) /μ = 55.4.

This Re value is selected to match one of the experimental cases in the paper by Hammad et al. (1999).

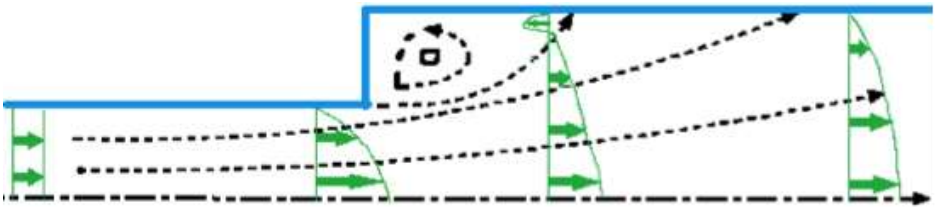
Use Fluent via Ansys Workbench to predict the flow and validate your results by comparing them with those in the following journal papers:

1. Macagno, E. and Hung, T-K.“Computational and experimental study of a captive annular eddy”, **J. Fluid Mech**. **vol. 28** (1967) pp. 43-64.
2. Hammad, KJ, Otugun, MV and Arik, EB “A PIV study of the laminar axisymmetric sudden expansion flow”, **Experiments in Fluids** 26 (1999) pp. 266-272.

Pre-Analysis & Start-Up — Lesson 2

Preliminary Analysis

The computational domain consists of two parts: one for the small pipe and the other for the larger pipe. In the small pipe, the viscous boundary layer grows along the pipe wall starting at the inlet, and eventually a fully developed velocity profile forms, provided that the small pipe is long enough. As that flow passes through the expansion entrance, a recirculation zone forms in the corner of the larger pipe. The flow will become fully developed again after sufficient distance downstream from the recirculation zone.

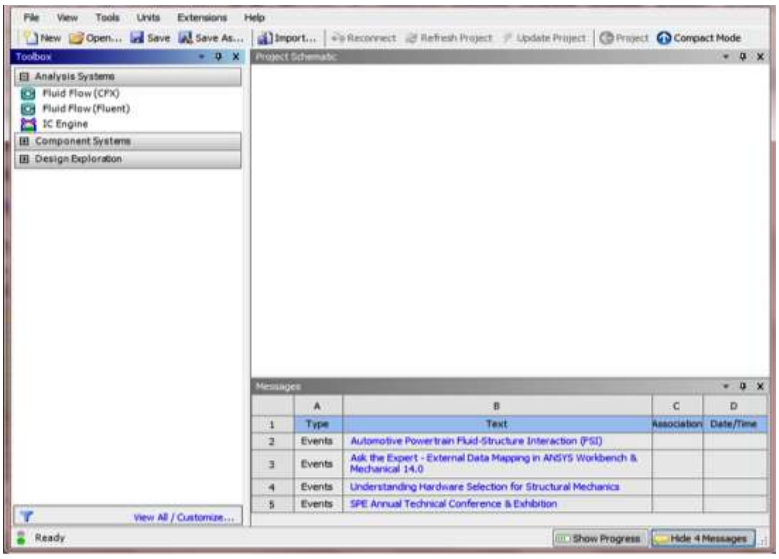


We'll create the geometry and mesh in Ansys software that is the pre-processor for Fluent, and then read the mesh into Fluent.

Start Ansys Fluent

Prior to opening the Ansys Workbench, create a folder called *pipe* in a convenient location. We'll use this as the working folder in which files created during the session will be stored. For this simulation Fluent will be run within the Ansys Workbench Interface. Start Ansys workbench: ***Start> All Programs > Ansys > Workbench***

The following figure shows the Workbench window.



Geometry — Lesson 3

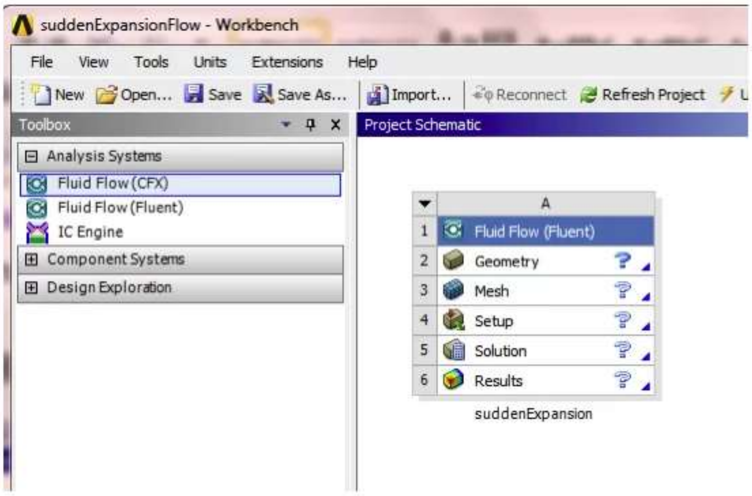
Saving

First, save the project at this point. Click on the "Save As…" button, which is located at the top of the ***Workbench Project Page***. Save the project as "suddenExpansionFlow" in your working directory. When you save in Ansys software a file and a folder will be created. For instance, if you save as "suddenExpansionFlow", a "suddenExpansionFlow" file and a folder called "suddenExpansionFlow_files" will appear. To reopen the Ansys files in the future you will need both the ".wbpj" file and the folder. If you do not have BOTH, you will not be able to access your project.

Fluid Flow (Fluent) Project Selection

On the left-hand side of the Workbench window, you will see a toolbox full of various analysis systems. To the right, you see an empty workspace. This is the place where you will organize your project. At the bottom of the window, you see messages from the Ansys software.

Left click (and hold) on ***Fluid Flow (Fluent)***, and drag the icon into the empty space in the ***Project Schematic***. Your Ansys Workbench window should now look comparable to the image below.



Since we selected Fluid Flow (Fluent), each cell of the system corresponds to a step in the process of performing CFD analysis using Fluent. Rename the project to *suddenExpansion*. We will work through each step from top down to obtain the solution to our problem.

Analysis Type

In the ***Project Schematic*** of the Workbench window, right click on ***Geometry*** and select ***Properties***, as shown below.

The properties menu will then appear to the right of the Workbench window. Under ***Advanced Geometry Options***, change the ***Analysis Type*** to 2D as shown in the image below.

Properties of Schematic A2: Geometry

	A	B
1	Property	Value
2	General	
3	Cell ID	Geometry
4	Geometry Source	
5	Geometry File Name	
6	Basic Geometry Options	
7	Solid Bodies	☑
8	Surface Bodies	☑
9	Line Bodies	☐
10	Parameters	☑
11	Parameter Key	DS
12	Attributes	☐
13	Named Selections	☐
14	Material Properties	☐
15	Advanced Geometry Options	
16	Analysis Type	2D
17	Use Associativity	☑
18	Import Coordinate Systems	☐
19	Import Work Points	☐
20	Reader Mode Saves Updated File	☐
21	Import Using Instances	☑
22	Smart CAD Update	☐
23	Enclosure and Symmetry Processing	☑
24	Mixed Import Resolution	None

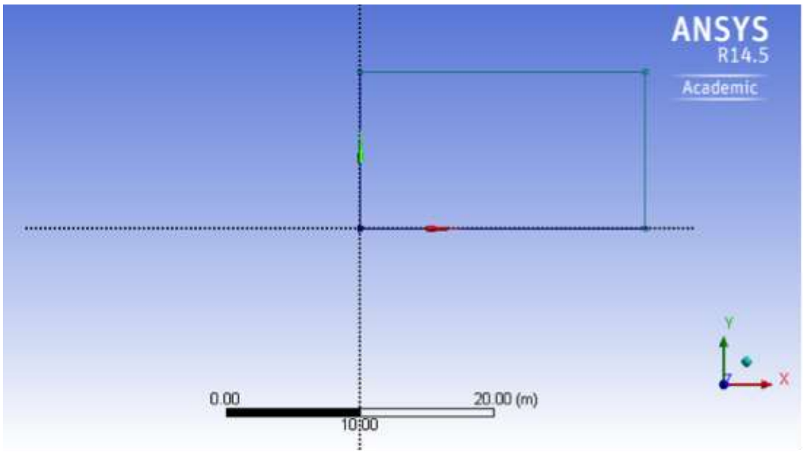
Launch Ansys DesignModeler
In the ***Project Schematic*** , double click on ***Geometry*** to start preparing the geometry.

At this point, a new window, Ansys Design Modeler will be opened. You will be asked to select a length unit. Use the default meter unit and click ***OK*** .

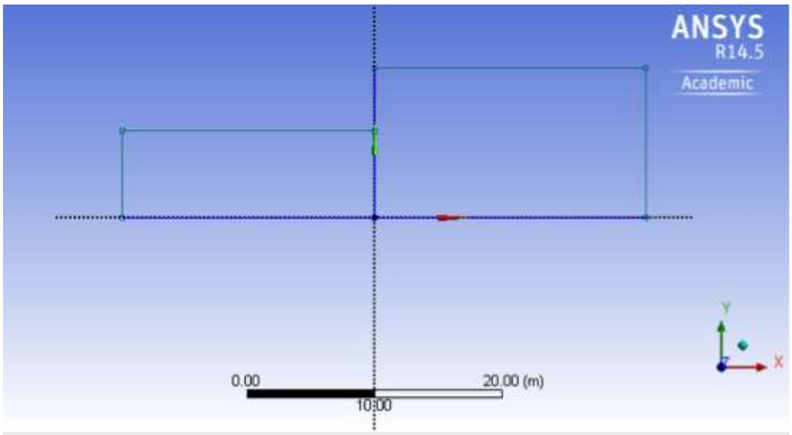
Creating a Sketch
Start by creating a sketch on the ***XYPlane*** . Under ***Tree Outline*** , select ***XYPlane*** , then click on ***Sketching*** right before ***Details View*** . This will bring up the ***Sketching Toolboxes*** .

Click on the ***+Z*** axis on the bottom right corner of the ***Graphics*** window to have a normal look of the XY Plane.

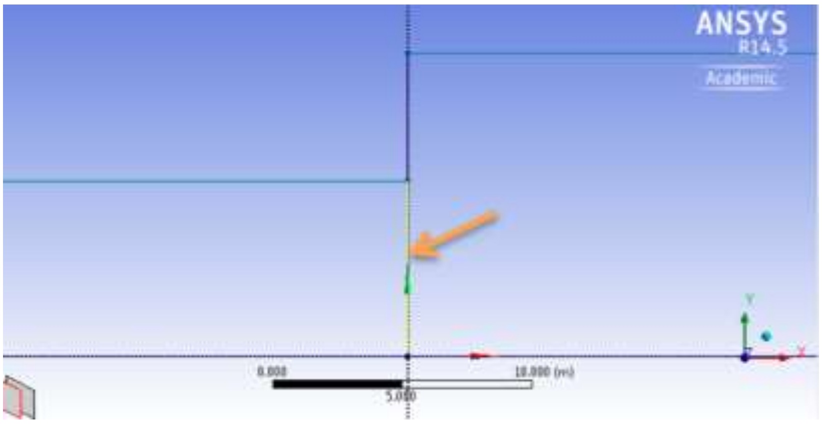
In the Sketching toolboxes, select ***Rectangle.*** In the ***Graphics*** window, create the *first rough Rectangle* by clicking once on the origin and then by clicking once somewhere in the positive XY plane. (Make sure that you see a letter P at the origin before you click. The P implies that the cursor is directly over a point of intersection.)



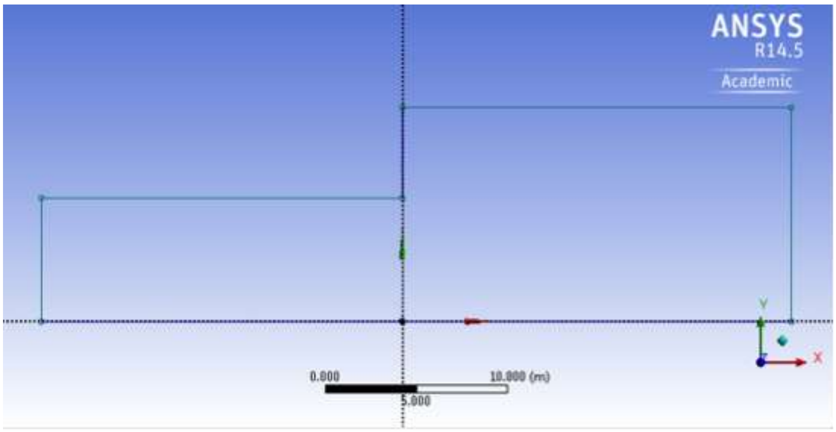
Repeat, and create the *second rectangle* with smaller length and width. Also, make sure these two rectangles are connected. At this point you should have something comparable to the image below.



The overlapped lines of these two rectangles are extraneous and will be removed. Under ***Sketching Toolboxes***, click ***Modify*** and ***cut*** the line indicated by the figure below. This will delete the width of the smaller rectangle.



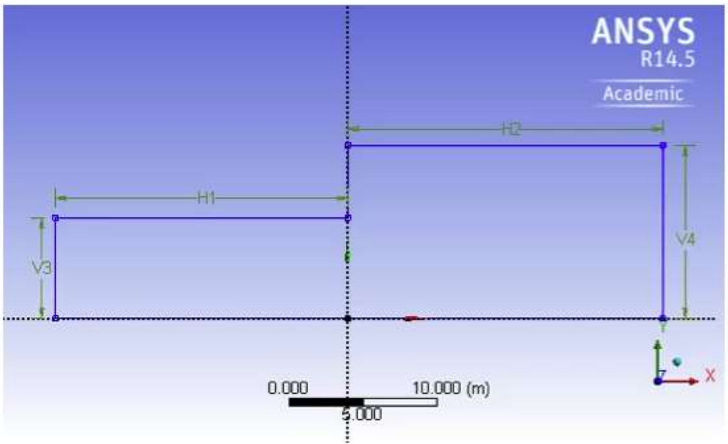
Then, ***trim*** the width (only the lower part) of the lager rectangle which overlaps the small rectangle. You should have the image below.



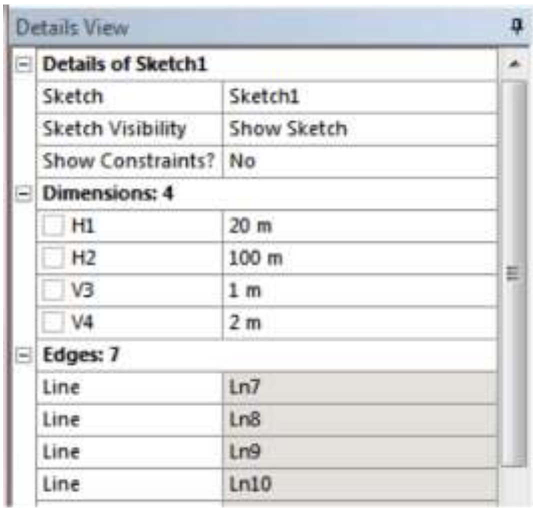
Dimensions

At this point the rectangle will be properly dimensioned.

Under ***Sketching Toolboxes*,** select ***Dimensions*** tab, use the default dimensioning tools. Dimension the geometry as shown in the following image.



Under the ***Details View*** table (located in the lower left corner), set H1=20m, H2=100m, V3=1m and V4 = 2m, as shown in the image below.

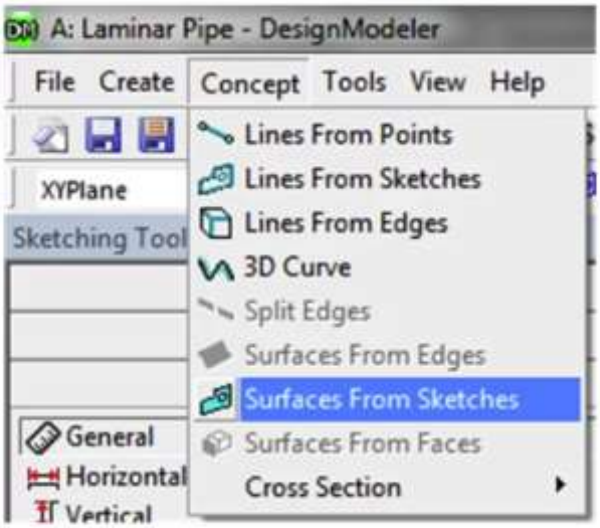


Details View

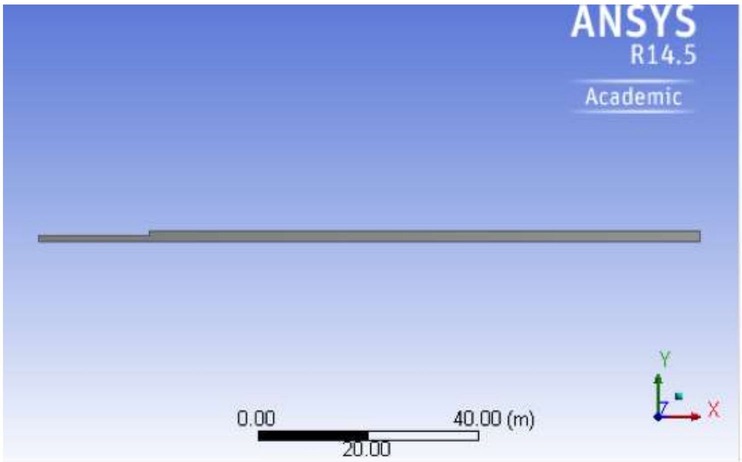
Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 4	
H1	20 m
H2	100 m
V3	1 m
V4	2 m
Edges: 7	
Line	Ln7
Line	Ln8
Line	Ln9
Line	Ln10

Surface Body Creation

To create the surface body, first ***(Click)Concept > Surface From Sketches*** as shown in the image below.



This will create a new surface ***SurfaceSK1***. Under ***Details View***, select ***Sketch1*** (go to ***XYplane*** and expand it to see ***Sketch1***, click on it) as ***Base Objects***, and then under Surface body select the thickness to 0.1m and click ***Apply***. Finally, right click ***SurfaceSK1*** and select ***Generate*** to generate the surface. You should see "***1 part, 1 Body*** " under ***SurfaceSK1***.



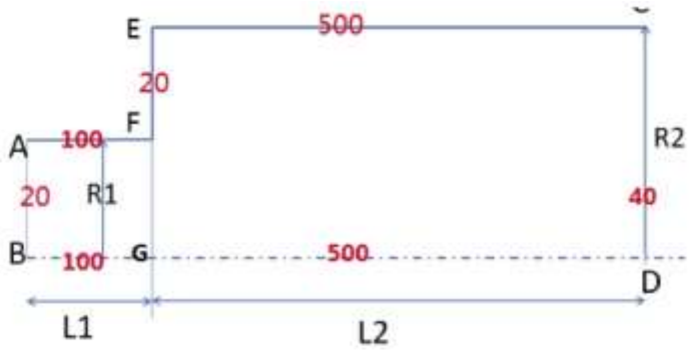
At this point, you can close *DesignModeler* and go back to ***Workbench Project Page***. You should see two green check marks next to ***Geometry*** and ***Mesh***. Press ***Refresh Project*** and ***Update Project***, then save your work thus far in the ***Workbench Project Page***.

For users of Ansys 15.0, please check the following topic for procedures for turning on the Auto Constraint feature before creating sketches in DesignModeler.

Mesh — Lesson 4

Sudden Expansion in Laminar Flow Mesh — Lesson 4

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:

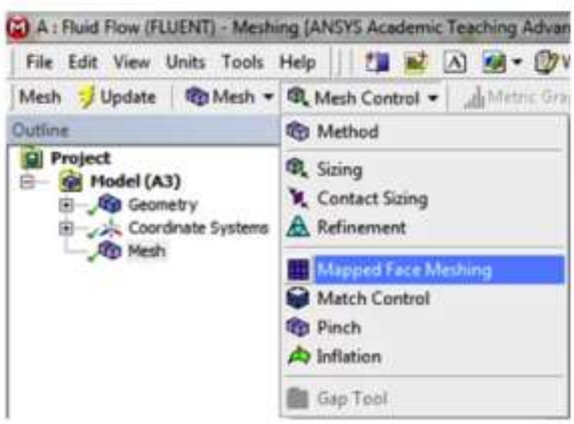
Launch Mesher

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.***"

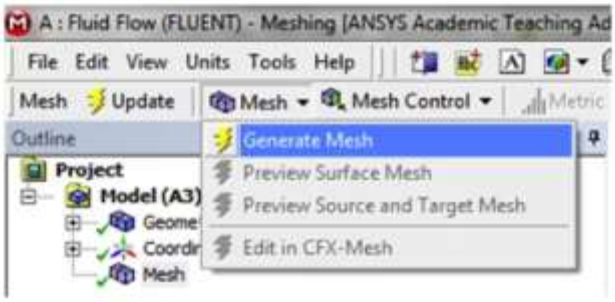
Mapped Face Meshing

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***. To incorporate this meshing style ***(Click) Mesh Control > Mapped Face Meshing*** as can be seen below.



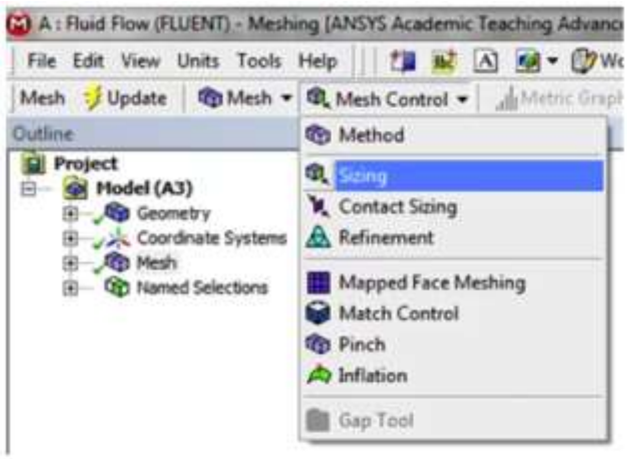
Now, the ***Mapped Face Meshing*** still must be applied to the pipe geometry. To do so, first click on the pipe body which should then highlight green. Next, ***(Click) Apply*** in the ***Details of Mapped Face Meshing*** table, as shown below.

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

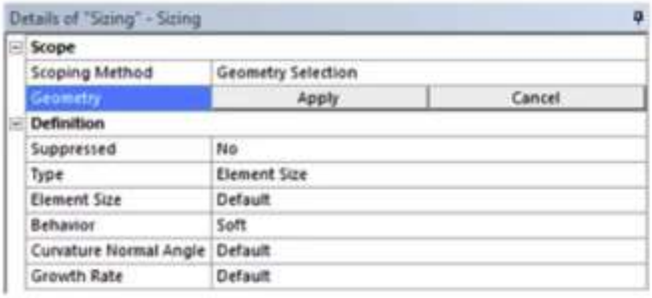


Edge Sizing

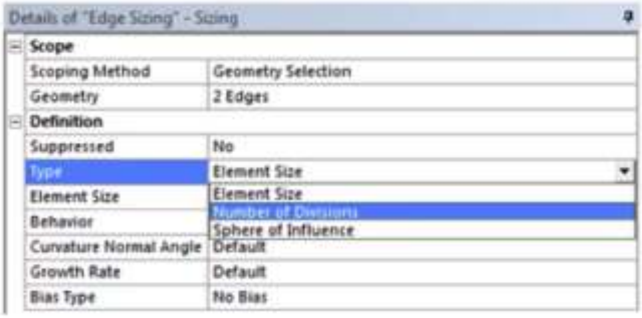
The desired mesh has specific number of divisions along the radial and the axial direction. To obtain the specified number of divisions ***Edge Sizing*** must be used. The divisions along the axial direction will be specified first. Now, an ***Edge Sizing*** needs to be inserted. First, ***(Click) Mesh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



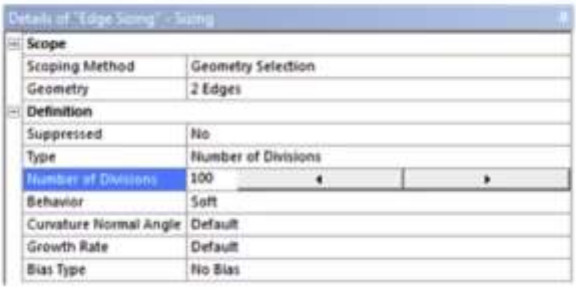
Now, the geometry and the number of divisions need to be specified. First ***(Click) Edge Selection Filter***, , in the upper toolbar. Hold down the "Ctrl" button and then click the edges ***AF*** and ***BG***. Both sides should highlight green. Next, hit ***Apply*** under the ***Details of Sizing***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



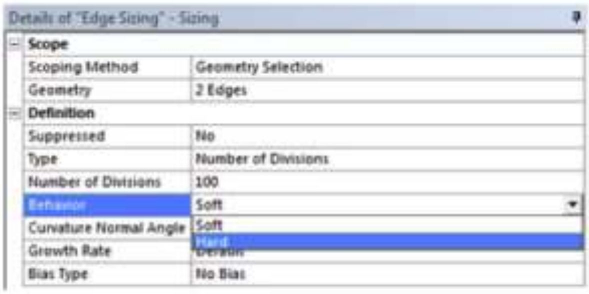
Now, change ***Type*** to ***Number of Divisions*** as shown in the image below.



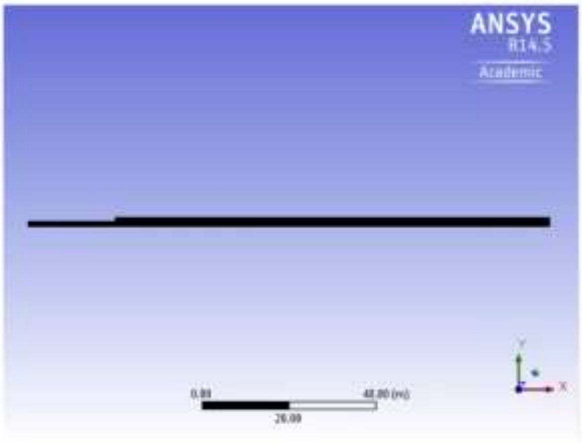
Then, set ***Number of Divisions*** to 100 as shown below.



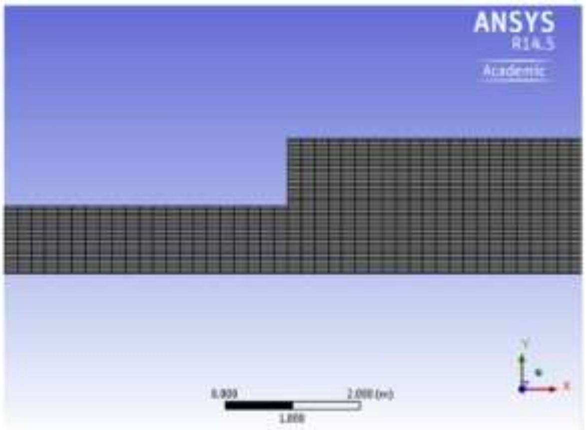
Change ***Behavior*** to ***Hard*** for both ***Edge Sizing's***.



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.

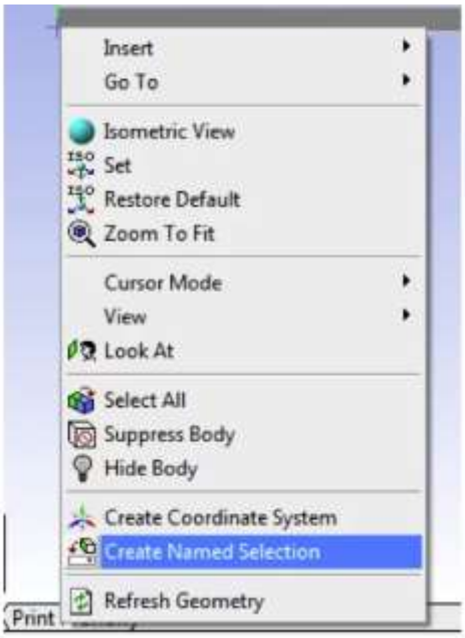


Mesh statistics can be found by clicking on ***Mesh*** in the tree and then by expanding ***Statistics*** under the ***Details of Mesh*** table. There are 22000 elements in the mesh above.

Create Named Selections

Here, the edges of the geometry will be given names so you 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.

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.



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

Now, create named selections for the remaining edges and name them according to the diagram. Since the “Centerline” consists of two edges, 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*** button, .

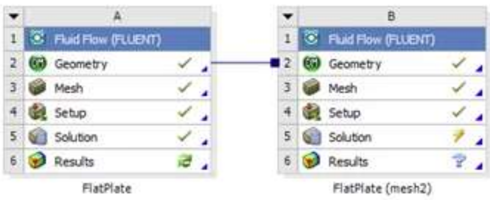
Verification & Validation — Lesson 8

Note: 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 total number of elements (cells) by a factor of four. To accomplish this, we double the number of divisions on each section. Instead of modifying the project that was just created, we will duplicate it and modify the duplicate. In the ***Workbench Project Page*** right click on ***Mesh*** then click ***Duplicate*** as shown below.

Rename the duplicate project to ***FlatPlate (mesh 2)***. You should have the following two projects in your ***Workbench Project Page***.

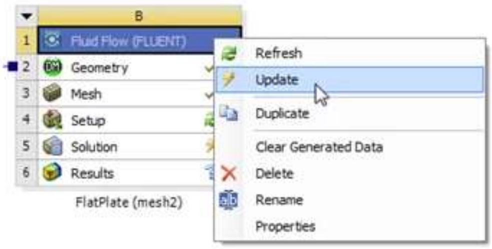


Next, double click on the ***Mesh*** cell of the ***FlatPlate (mesh 2)*** project. A new Ansys Mesher window will open. Under ***Outline***, expand ***Mesh*** and click on ***Edge Sizing***. Under Details of "Edge Sizing," enter 100 for ***Number of Divisions***. Next, set the number of divisions for ***Edge Sizing 2*** and ***Edge Sizing 3*** to 120. Then, click ***Update*** to create the new mesh. The new mesh should now have 12,000 elements (100 x 1200). A quick glance of the mesh statistics reveals that there are, indeed, 12,000 elements.

Statistics	
Nodes	12221
Elements	12000
Mesh Metric	None

Compute the Solution

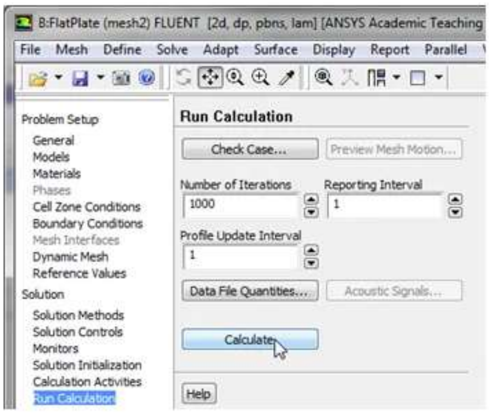
Close the Ansys Mesher to go back to the ***Workbench Project Page***. Under ***FlatPlate (mesh 2)***, right click on ***Fluid Flow (Fluent)*** and click on ***Update***, as shown below.



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

Convergence

To launch Fluent, double click on the ***Solution*** of the "FlatPlate (mesh 2)" project in the ***Workbench Project Page***. The new mesh has significantly more cells, thus it is likely that the solution did not converge to the tolerances we have previously set. Therefore, we will iterate the solution further, to make sure that the solution converges. To do so, click on ***Run Calculation***, set ***Number of Iterations*** to 1000 and click ***Calculate***, as shown below.

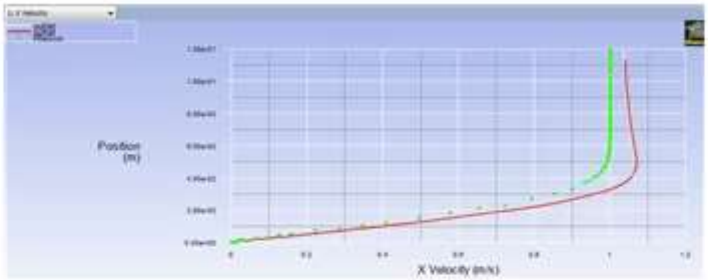


Once you rerun the calculation, you will quickly see that the solution did not converge for the finer mesh within 1000 iterations. The solution should converge by iteration 1784, as shown below.

```
! 1784 solution is converged
  1784 9.9753e-07 1.7521e-09 3.9381e-10  0:00:11   216
```

Outlet Velocity Profile

Now, the variation of the x component of the velocity will be plotted with the results of the original mesh to determine whether the solution is mesh converged. Set up a plot for the variation of the x component of the velocity along the outlet as was done in the <u>solution</u> section. Then load the *XVelOutlet.xy* file into the plot and generate the plot. You should obtain the following image.



As you can see from the image above, the numerical solution does not vary much between the two meshes. Thus, it has been confirmed that the solution is mesh converged.