Laminar Flow Through Sudden Expansion in an Axisymmetric Pipe

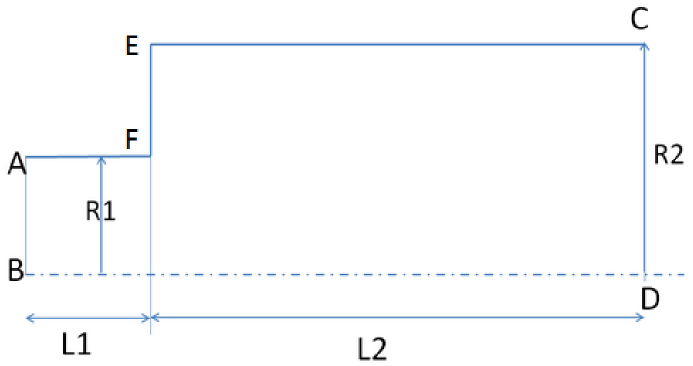
Tutorial

Yong Wang & S. Elghobashi

Mech. and Aerospace Eng. Dept. UCI

August 2013

Problem Specification



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, $U1$ = 1. m/s. The fluid exhausts into the ambient atmosphere which is at a pressure of 1 atm at CD. The density $\rho = 1kg/m^3$, and the dynamic viscosity is:

$\mu = 3.61x\ 10^{-2} kg/(ms).$

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:

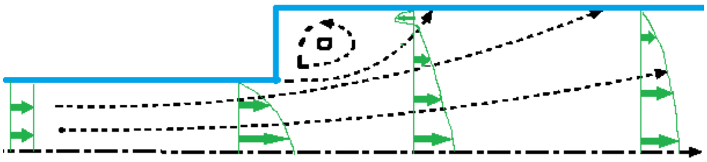
1- Macagno, E. and Hung, T-K. "Computational and experimental study of a captive annular eddy", **J. Fluid Mech**. **vol. 28** (1967) pp. **43-64.**

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.

Step 1: Pre-Analysis & Start-up

Preliminary Analysis

The computational domain consists of 2 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.

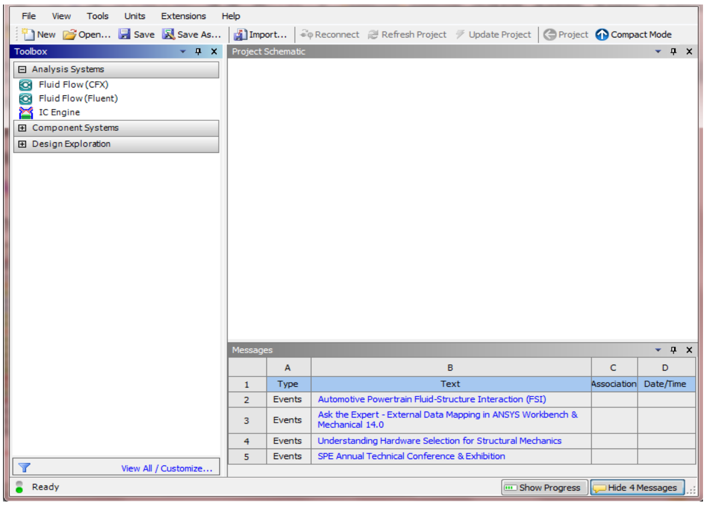


We'll create the geometry and mesh in ANSYS 14.0 which is the preprocessor for FLUENT, and then read the mesh into FLUENT.

Start ANSYS FLUENT

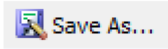
Prior to opening ANSYS, create a folder called *pipe* in a convenient location. We'll use this as the working folder in which files created during the session will be stored. For this simulation FLUENT will be run within the ANSYS Workbench Interface. Start ANSYS workbench: ***Start> All Programs> Ansys 14.*> Workbench 14.****

The following figure shows the workbench window.



Step 2: Geometry

Saving

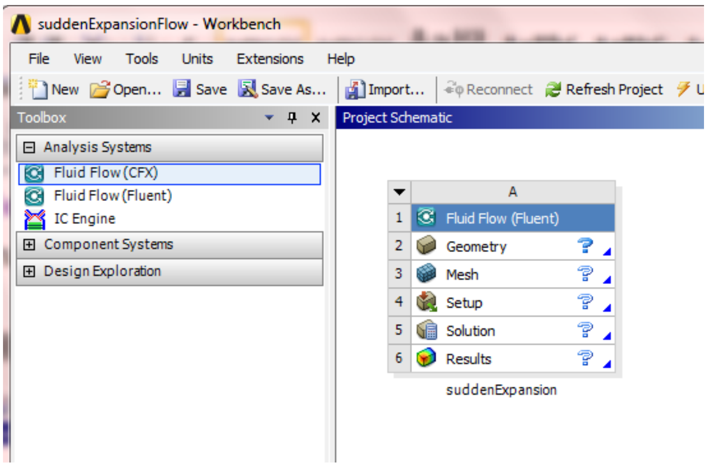
First, save the project at this point. Click on the "Save As.." button, ,

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 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. In order 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 work space. This is the place where you will organize your project. At the bottom of the window, you see messages from ANSYS.

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

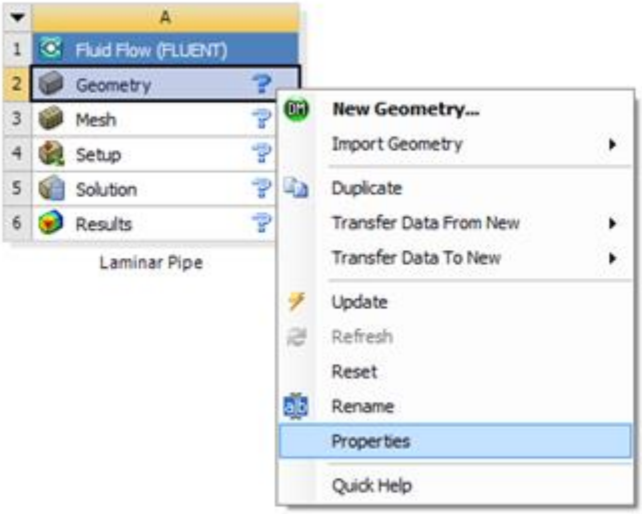


Since we selected Fluid Flow (FLUENT), each cell of the system corresponds to a step in the process of performing CFD analysis using FLUENT. Rename the project to *suddenExpansion*.

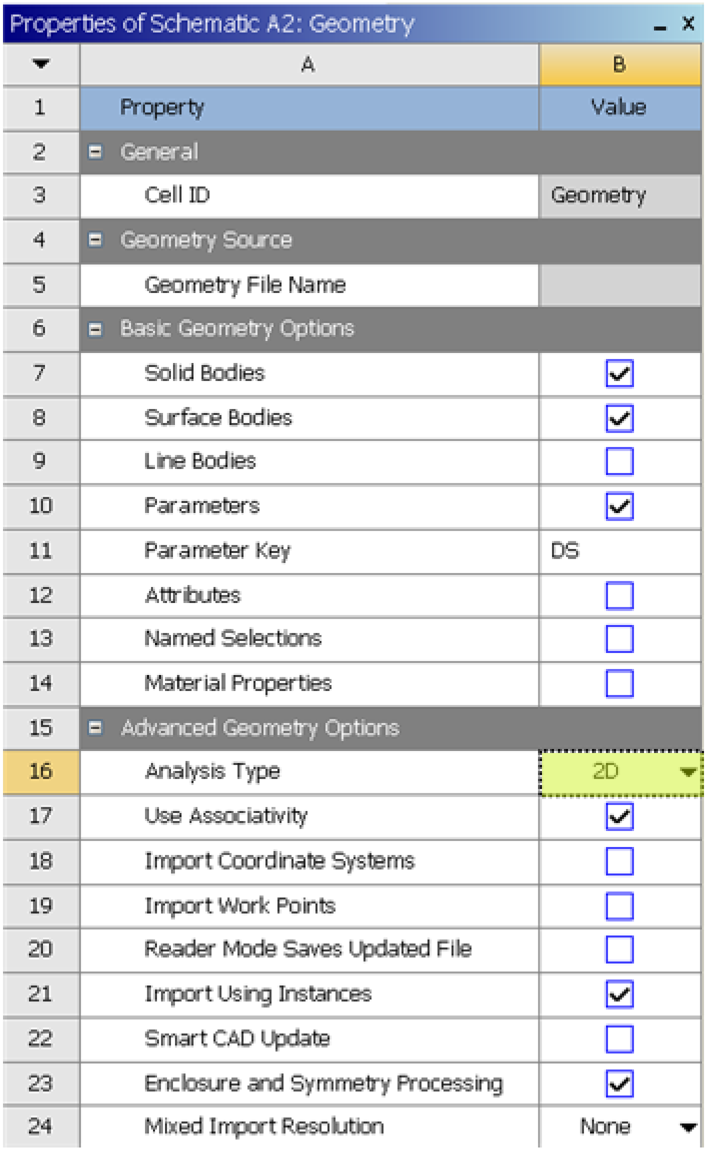
We will work through each step from top down to obtain the solution to our problem.

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 ***Advance 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 Design Modeler

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

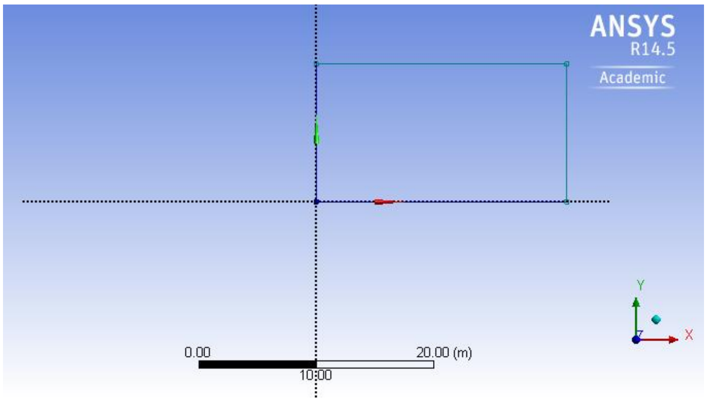
At this point, a new window, ANSYS Design Modeler will be opened. You will be asked to select desired length unit. Use the default meter unit and click ***OK***.

Creating a Sketch

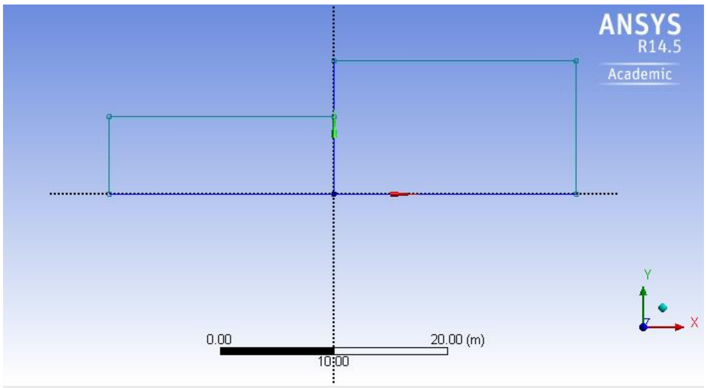
Start by creating a sketch on the ***XYPlane***. Under ***Tree Outline***, select ***XYPlane***, then click on ***Sketching*** right before ***Details View***. This will bring up the ***Sketching Toolboxes***.

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

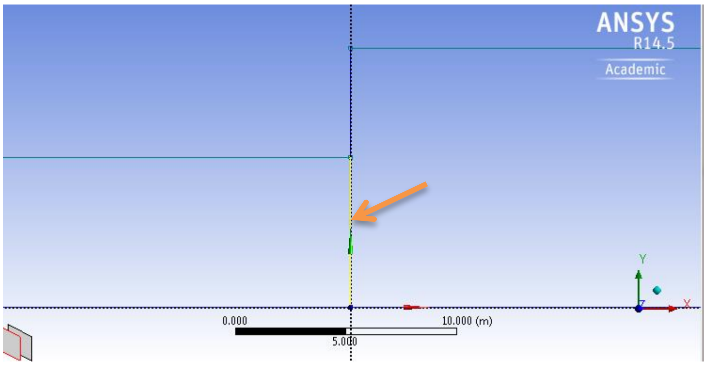
In the Sketching toolboxes, select ***Rectangle***. In the ***Graphics*** window, create the *first rough Rectangle* by clicking once on the origin and then by clicking once somewhere in the positive XY plane. (Make sure that you see a letter P at the origin before you click. The P implies that the cursor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



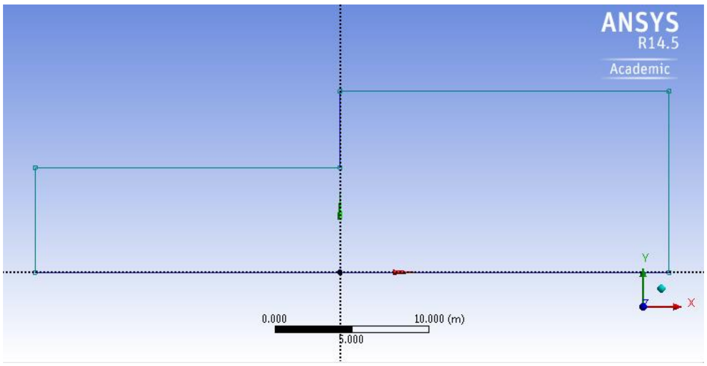
Repeat, and create the *second rectangle* with smaller length and width. Also, make sure these two rectangles are connected. At this point you should have something comparable to the image below.



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



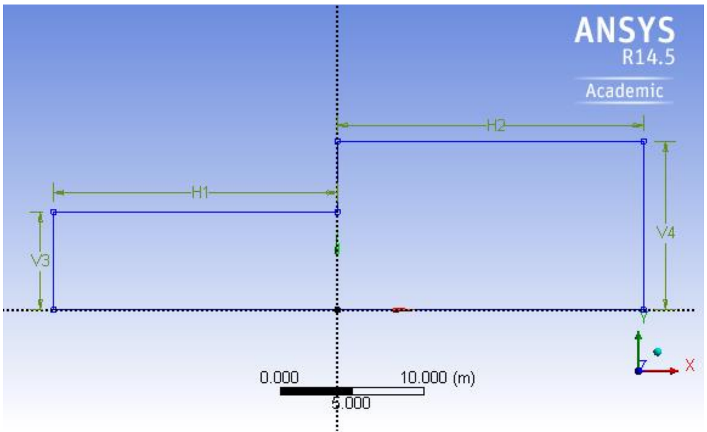
Then, ***trim*** the width (only the lower part) of the lager rectangle which overlaps the small rectangle. You should have the image below.



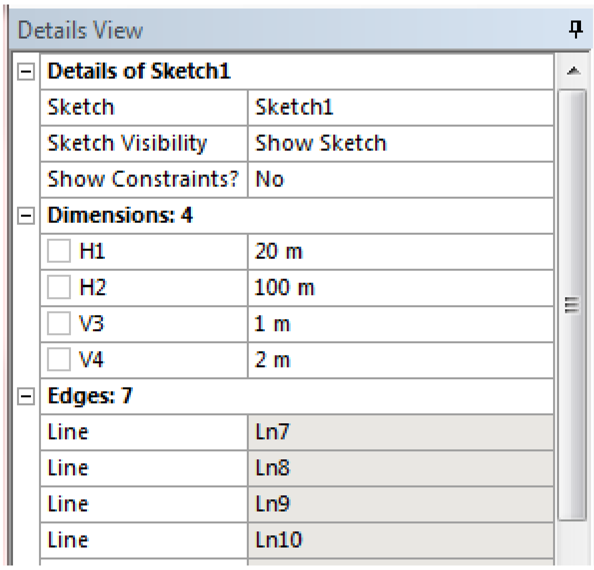
Dimensions

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.

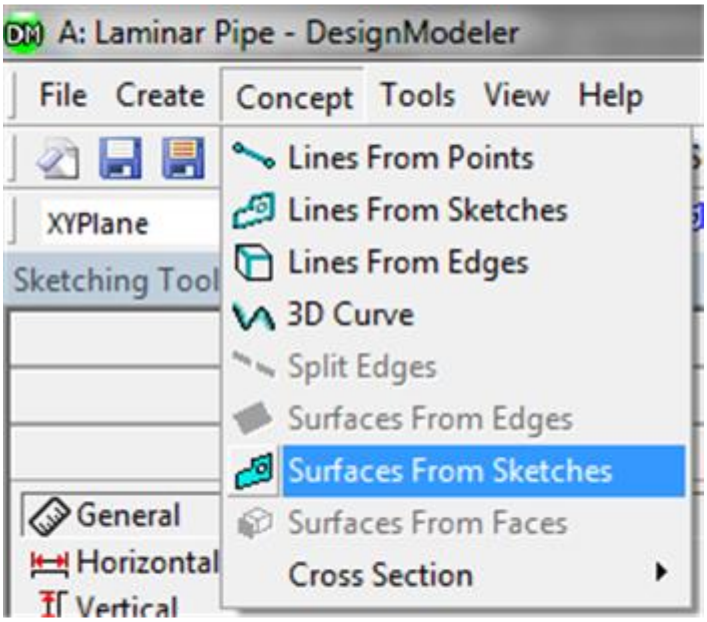


Under the ***Details View*** table (located in the lower left corner), set H1=20m, H2=100m, V3=1m and V4 = 2m, as shown in the image below.

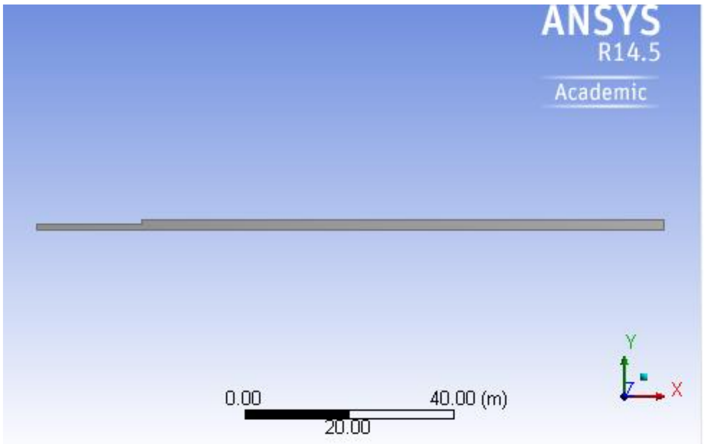


Surface Body Creation

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



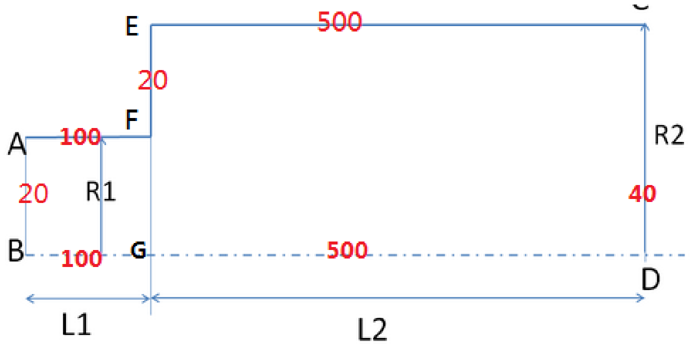
This will create a new surface ***SurfaceSK1***. Under ***Details View***, select ***Sketch1*** (go to ***XYplane*** and expand it to see ***Sketch1***, click on it) as ***Base Objects***, and then under Surface body select the thickness to 0.1m and click ***Apply***. Finally right click ***SurfaceSK1***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



At this point, you can close the ***Design Modeler*** 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***.

Step 3: Mesh

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



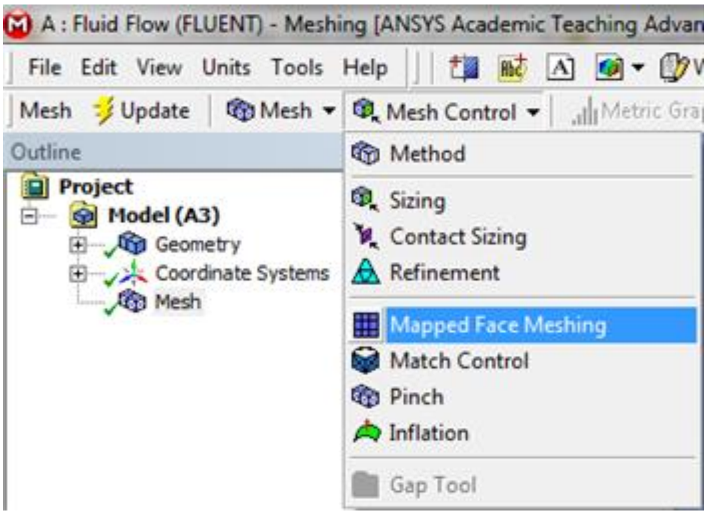
Launch Mesher

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

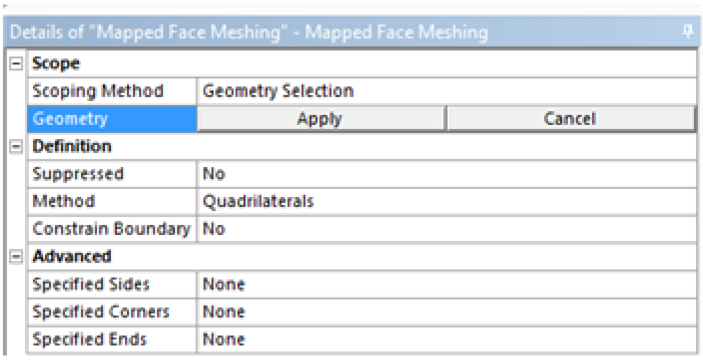
You may want to generate a preliminary mesh by right clicking on ***Mesh*** (upper window) and then "***Generate Mesh***".

Mapped Face Meshing

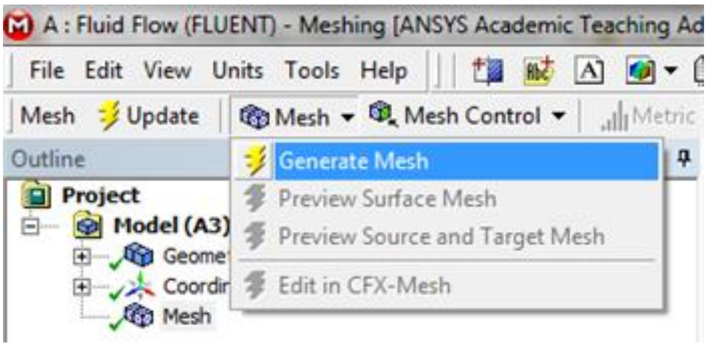
Here we are interested in creating a grid style of mesh that can be mapped to a rectangular domain. This meshing style is called ***Mapped Face Meshing***. In order to incorporate this meshing style ***(Click) Mesh Control > Mapped Face Meshing*** as can be seen below.



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.



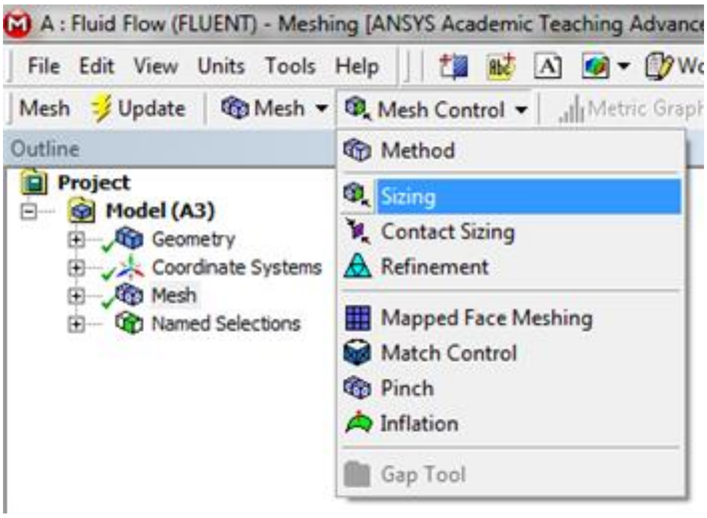
After clicking ***Apply***, you should see "***Geometry: 1Face***". Now, ***(Click) Mesh > Generate Mesh*** as can be seen below, and generate a rough mesh.



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.



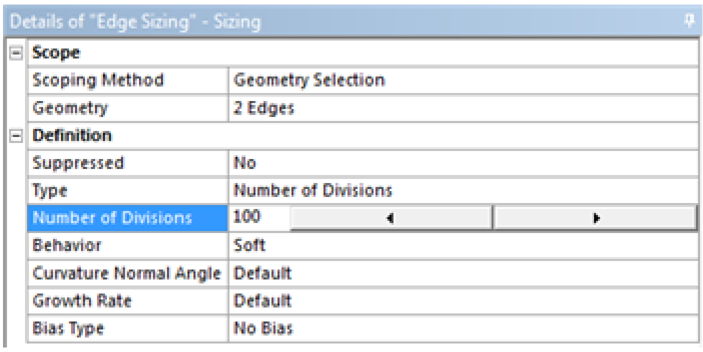
Now, the geometry and the number of divisions need to be specified. First ***(Click) Edge Selection Filter***, , in the upper toolbar. Hold down the "Ctrl" button and then click the edges ***AF*** and ***BG***. Both sides should highlight green. Next, hit ***Apply*** under the ***Details of Sizing***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

Details of "Sizing" - Sizing		
Scope		
Scoping Method	Geometry Selection	
Geometry	Apply	Cancel
Definition		
Suppressed	No	
Type	Element Size	
Element Size	Default	
Behavior	Soft	
Curvature Normal Angle	Default	
Growth Rate	Default	

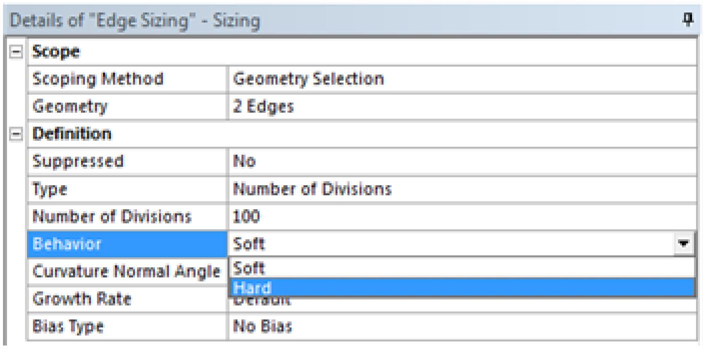
Now, change ***Type*** to ***Number of Divisions*** as shown in the image below.

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

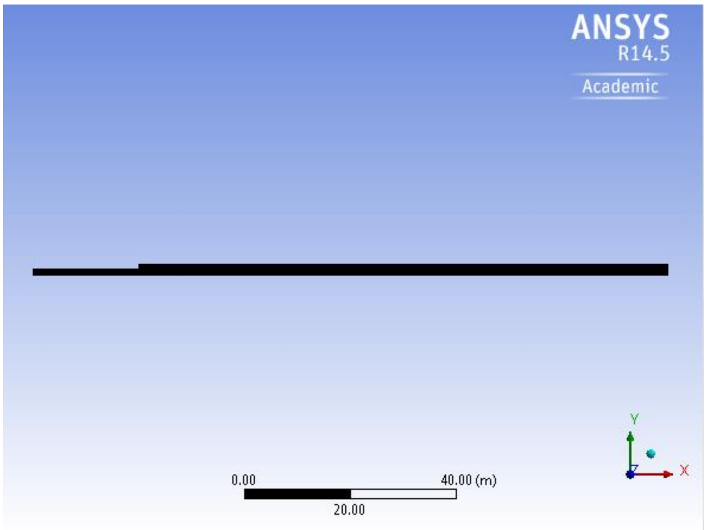
Then, set ***Number of Divisions*** to 100 as shown below.



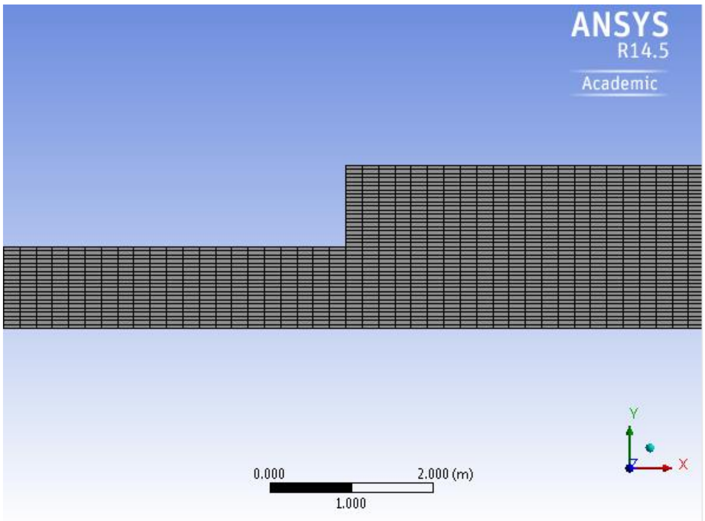
Change ***Behavior*** to ***Hard*** for both ***Edge Sizing's***.



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.



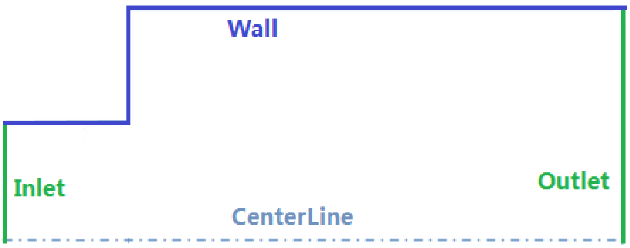
Detail around the expansion entrance FG is presented blow.



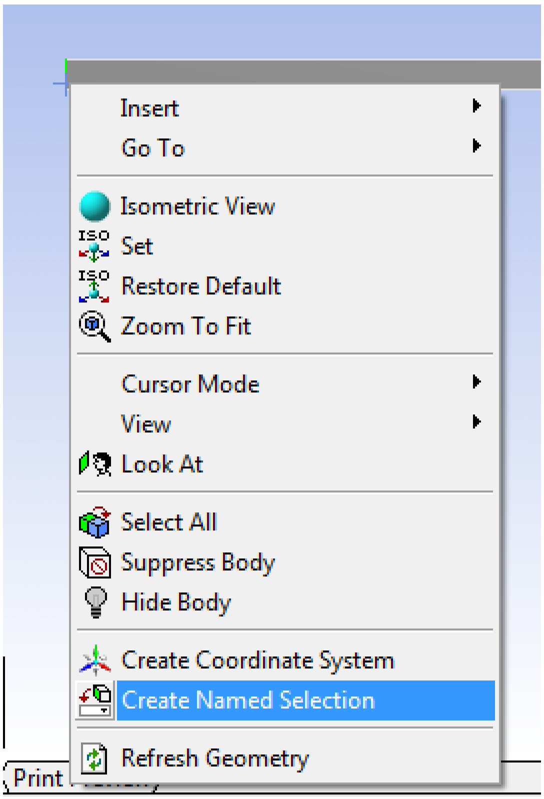
Mesh statistics can be found by clicking on ***Mesh*** in the tree and then by expanding ***Statistics*** under the ***Details of Mesh*** table. It can be seen that there are 22000 elements in the mesh above.

Create Named Selections

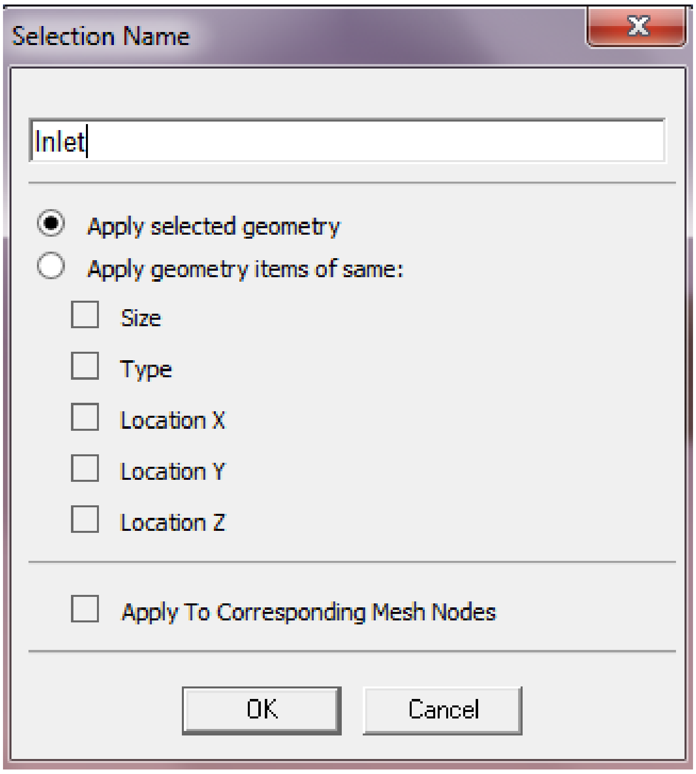
Here, the edges of the geometry will be given names so one can assign boundary conditions in Fluent in later steps. The left side of the pipe will be called "Inlet" and the right side will be called "Outlet". The bottom side will be called "CenterLine" and the other edges are called "Wall", as shown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



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.



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 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

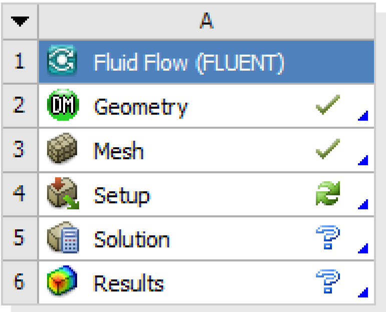
First save the project in the Mesher window. Next, close the Mesher window. Then, go to the ***Workbench Project Page*** and click the ***Update Project*** button, 

.

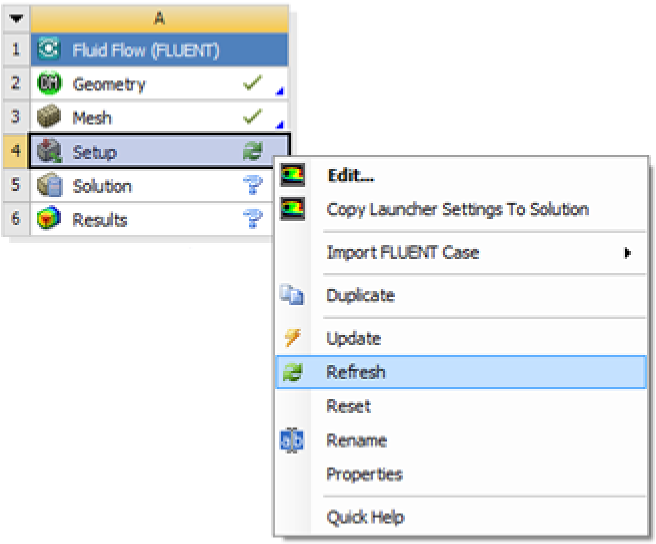
If the Progress window shows error message like this: "Model information is incompatible with incoming mesh", right click on ***Setup*** in the workbench and select "***Reset***", then ***Update Project*** again.

Step 4: Setup (Physics)

Your current ***Workbench Project Page*** should look comparable to the following image. Regardless of whether you downloaded the mesh and geometry files or if you created them yourself, you should have checkmarks to the right of ***Geometry*** and ***Mesh***.



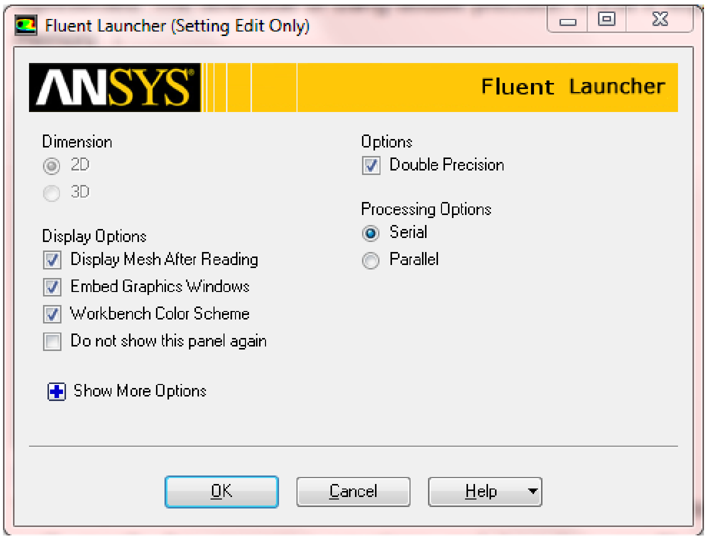
Next, the mesh and geometry data need to be read into FLUENT. To read in the data ***(Right Click) Setup > Refresh*** in the ***Workbench Project Page*** as shown in the image below.



After you click ***Update***, a question mark should appear to the right of the ***Setup*** cell. This indicates that the ***Setup*** process has not yet been completed.

Launch Fluent

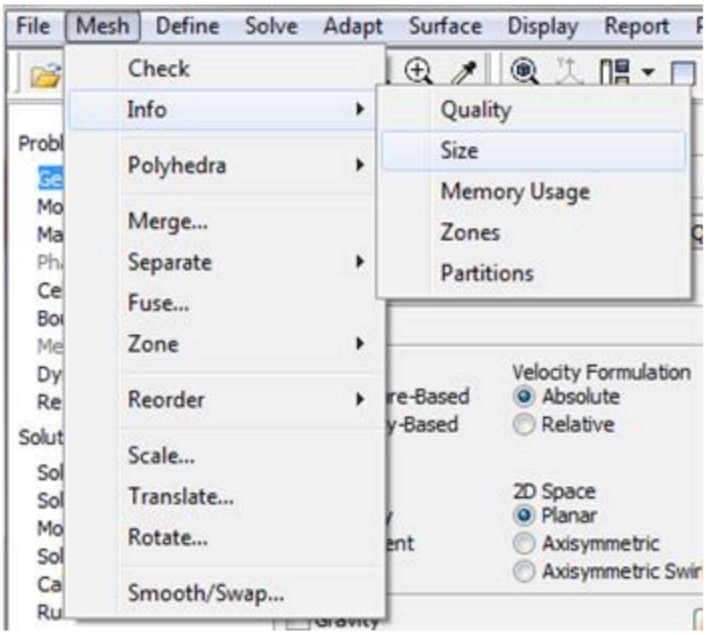
Double click on ***Setup*** in the ***Workbench Project Page*** which will bring up the ***FLUENT Launcher***. When the ***FLUENT Launcher*** appears change the options to "Double Precision", and then click ***OK*** as shown below. The Double Precision option is used to select the double-precision solver. In the double-precision solver, each floating point number is represented using 64 bits in contrast to the single-precision solver which uses 32 bits. The extra bits increase not only the precision, but also the range of magnitudes that can be represented. The downside of using double precision is that it requires more memory.



Twiddle your thumbs a bit while the FLUENT interface starts up. This is where we'll specify the governing equations and boundary conditions for our boundary-value problem. On the left-hand side of the FLUENT interface, we see various items listed under ***Problem Setup***. We will work from top to bottom of the ***Problem Setup*** items to setup the physics of our boundary-value problem. On the right hand side, we have the ***Graphics*** pane and, below that, the ***Command*** pane.

Check and Display Mesh

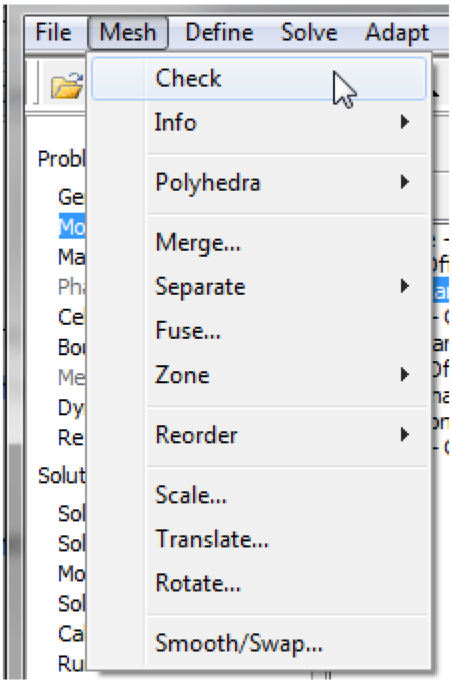
First, the mesh will be checked to verify that it has been properly imported from ***Workbench***. In order to obtain the statistics about the mesh ***(Click) Mesh > Info > Size***, as shown in the image below.



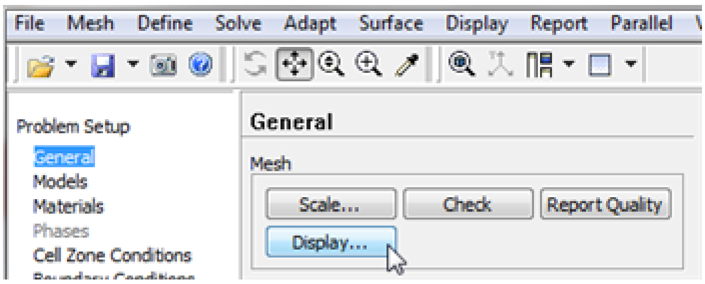
Then, you should obtain the following output in the ***Command*** pane.

```
Mesh Size

Level     Cells     Faces     Nodes    Partitions
    0     22000     44640     22641             1

 1 cell zone, 5 face zones.
```

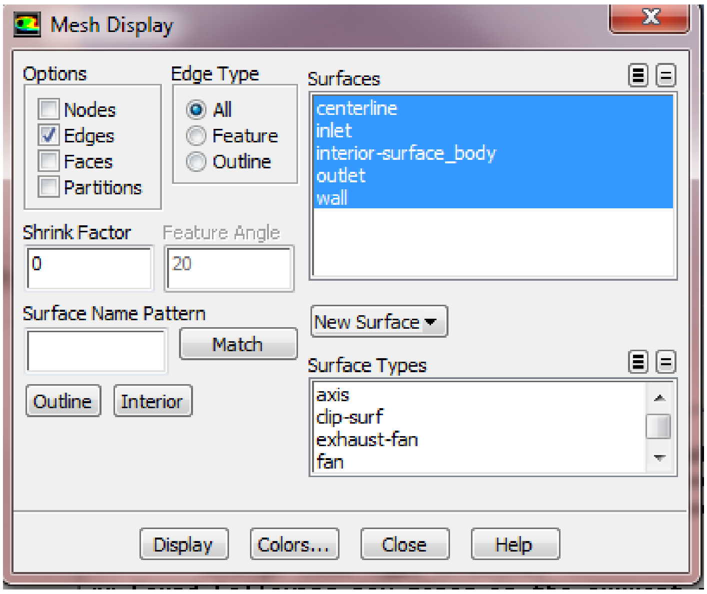
The mesh that was created earlier has 22000 elements. Note that in FLUENT elements are called cells. The output states that there are 22000 cells, which is a good sign. Next, FLUENT will be asked to check the mesh for errors. In order to carry out the mesh checking procedure ***(Click) Mesh > Check*** as shown in the image below.



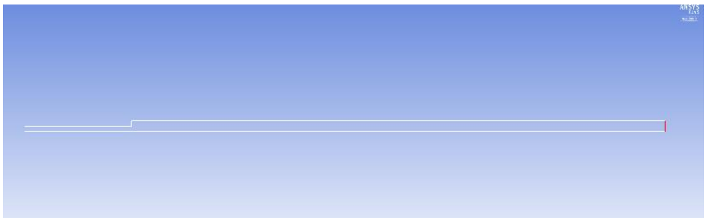
You should see no errors in the ***Command*** Pane. Now, that the mesh has been verified, the mesh display options will be discussed. In order to bring up the display options ***(Click) General > Mesh > Display*** as shown in the image below.



The previous step should cause the ***Mesh Display*** window to open, as shown below. Note that the ***Named Selections*** created in the meshing steps now appear.



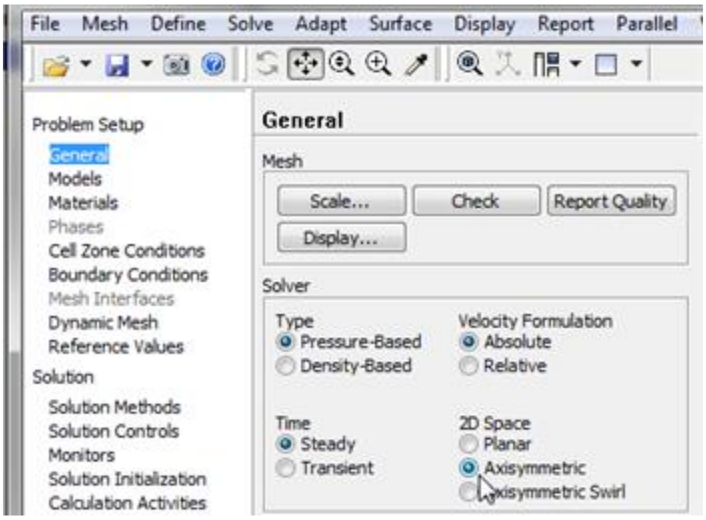
You should have all the surfaces shown in the above snapshot. Clicking on a surface name in the ***Mesh Display*** menu will toggle between select and unselect. Clicking ***Display*** will show all the currently selected surface entities in the graphics pane. Unselect all surfaces and then select each one in turn to see which part of the domain or boundary the particular surface entity corresponds to (you will need to zoom in/out and translate the model as you do this). For instance, if you select ***wall***, ***outlet***, and ***centerline*** and then click ***Display*** you should then obtain the following output in the graphics window.



Now, make sure all 5 items under ***Surfaces*** are selected. The  button next to ***Surfaces*** selects all of the boundaries while the  button deselects all of the boundaries at once. Once all the 5 boundaries have been selected click ***Display***, then close the ***Mesh Display*** window. The region displayed in the graphics window corresponds to our solution domain.

Define Solver Properties

In this section the various solver properties will be specified in order to obtain the proper solution for the laminar pipe flow. First, the axisymmetric nature of the geometry must be specified. Under ***General > Solver > 2D Space*** select ***Axisymmetric*** as shown in the image below.

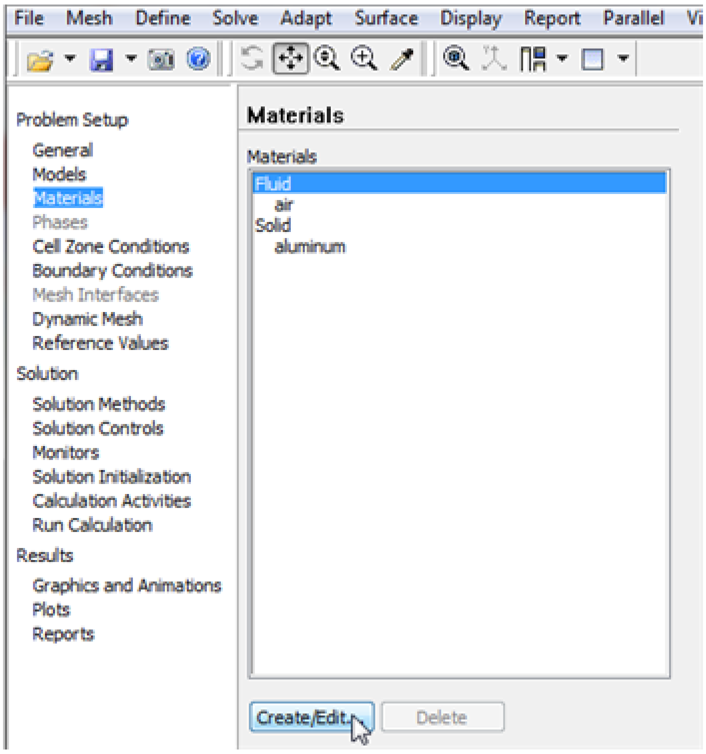


Next, the ***Viscous Model*** parameters will be specified. In order to open the Viscous Model Options ***Models > Viscous - Laminar > Edit...***. By default, the Viscous Model options are set to laminar, so no changes are needed. Click ***Cancel*** to exit the menu.

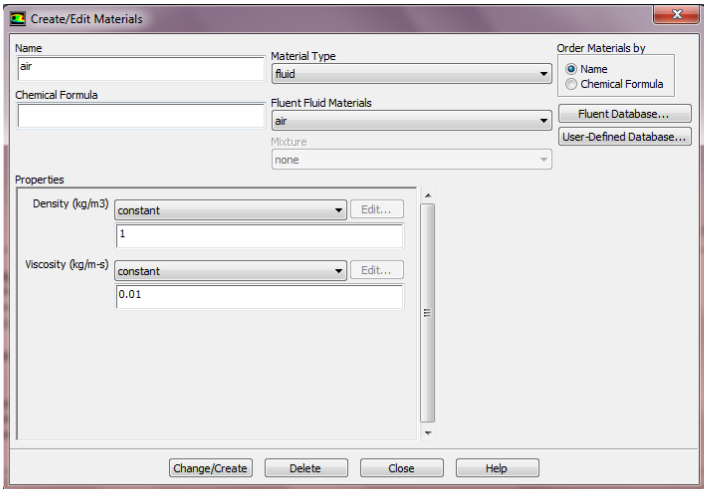
Now, the Energy Model parameters will be specified. In order to open the Energy Model Options ***Models > Energy-Off > Edit...***. For incompressible flow, the energy equation is decoupled from the continuity and momentum equations. We need to solve the energy equation only if we are interested in determining the temperature distribution. We will not deal with temperature in this example. So leave the ***Energy Equation*** set to off and click ***Cancel*** to exit the menu.

Define Material Properties

Now, the properties of the fluid that is being modeled will be specified. The properties of the fluid were specified in the Problem Specification section. In order to create a new fluid ***(Click) Materials > Fluid > Create/Edit...*** as shown in the image below.



In the ***Create/Edit Materials*** menu set the ***Density*** to 1kg/m^3 (constant) and set the ***Viscosity*** to 3.61e-2 kg/(ms) (constant) as shown in the image below.



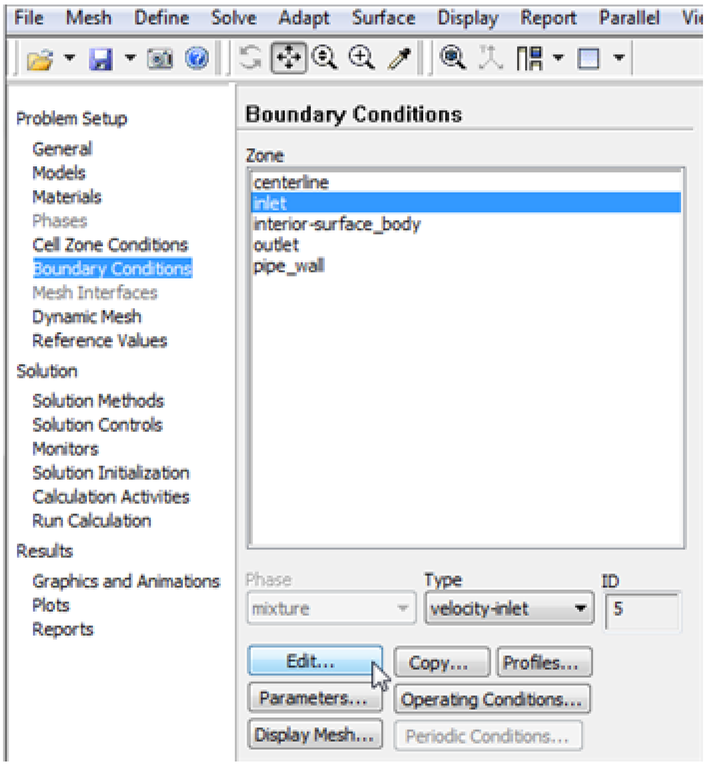
Click ***Change/Create***. Close the window.

Define Boundary Conditions

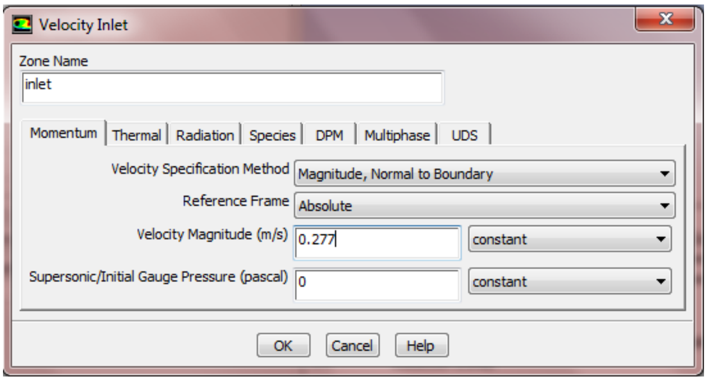
At this point the boundary conditions for the four ***Named Selections*** will be specified. The boundary condition for the *inlet* will be specified first.

Inlet Boundary Condition

In order to start the process ***(Click) Boundary Conditions > inlet > Edit…*** as shown in the following image.



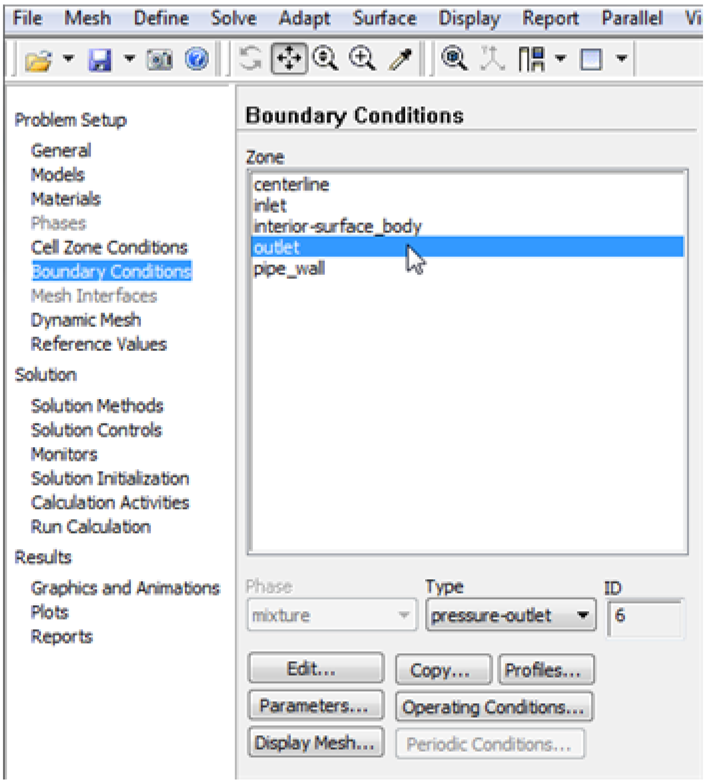
Note that the ***Boundary Condition Type*** should have been automatically set to ***velocity-inlet***. Now, the velocity at the ***inlet*** will be specified. In the ***Velocity Inlet*** menu set the ***Velocity Specification Method*** to ***Magnitude, Normal to Boundary***, and set the ***Velocity Magnitude (m/s)*** to 0.277 m/s, as shown below.



Then, click ***OK*** to close the ***Velocity Inlet*** menu.

Outlet Boundary Condition

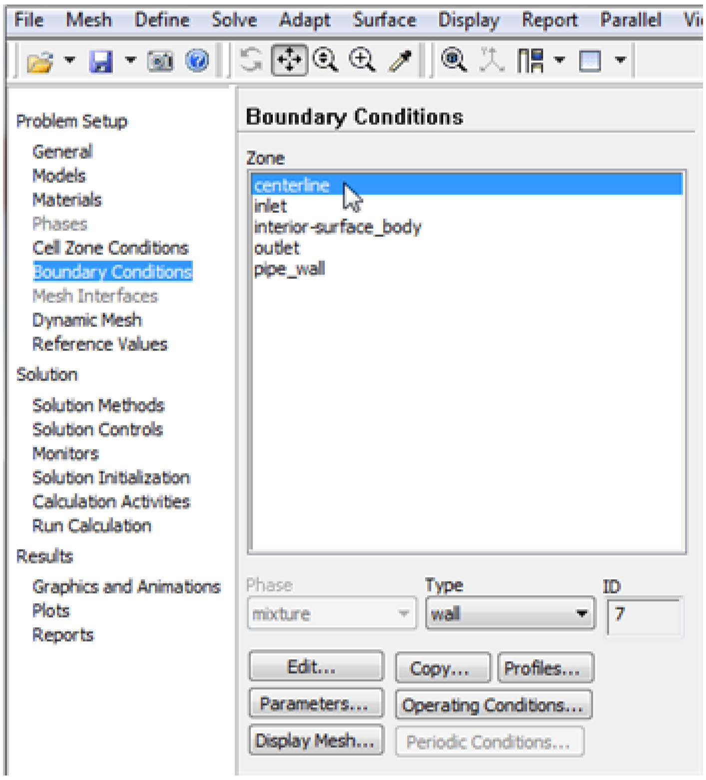
First, select *outlet* in the ***Boundary Conditions*** menu, as shown below.



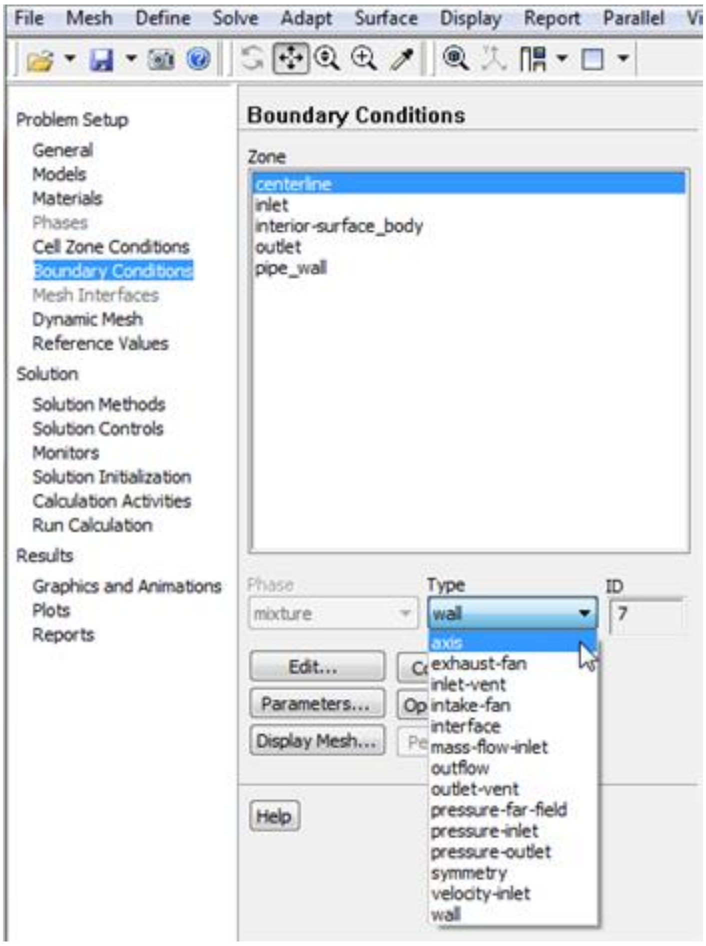
As can be seen in the image above the ***Type*** should have been automatically set to ***pressure-outlet***. If the ***Type*** is not set to ***pressure-outlet***, then set it to ***pressure-outlet***. Now, no further changes are needed for the *outlet* boundary condition.

Centerline Boundary Condition

Select *centerline* in the ***Boundary Conditions*** menu, as shown below.



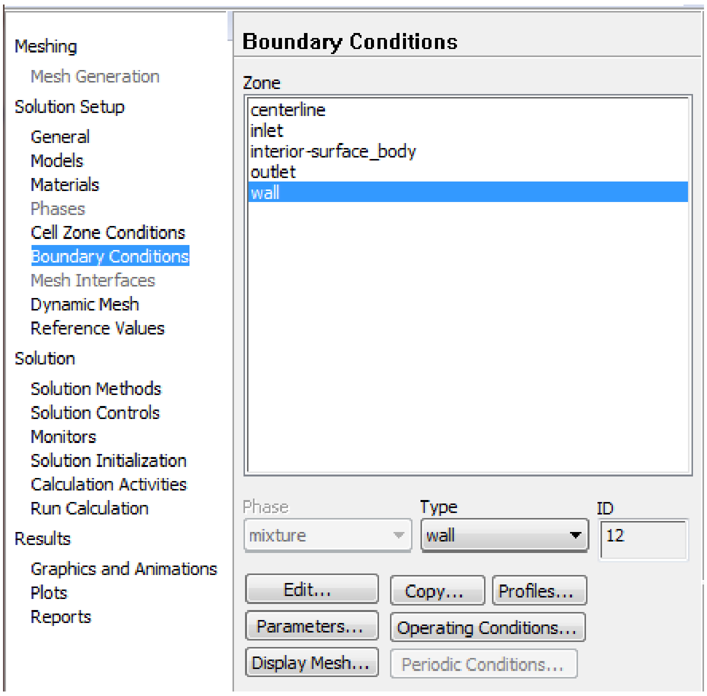
As can be seen in the image above the ***Type*** has been automatically set to ***wall*** which is not correct. Change the ***Type*** to ***axis***, as shown below.



When the dialog boxes appear click ***Yes*** to change the boundary type. Then click ***OK*** to accept "centerline" as the zone name.

Pipe Wall Boundary Condition

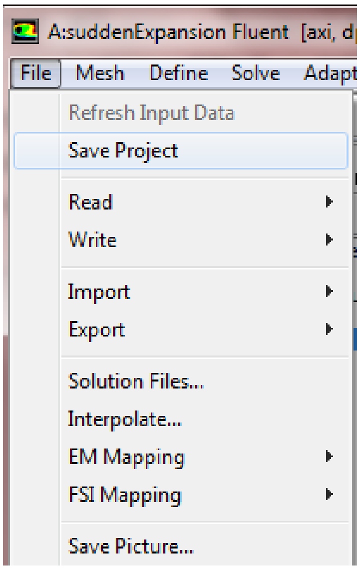
First, select *Wall* in the ***Boundary Conditions*** menu, as shown below.



As can be seen in the image above the ***Type*** should have been automatically set to ***wall***. If the ***Type*** is not set to ***wall***, then set it to ***wall***. Now, no further changes are needed for the *wall* boundary condition.

Save

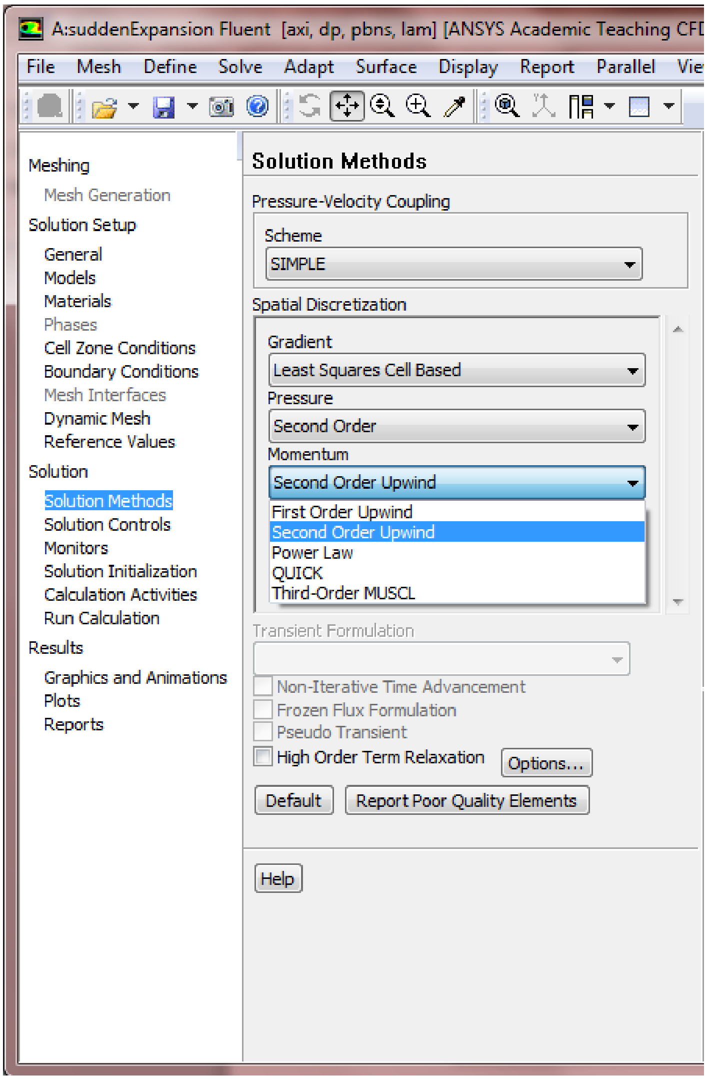
In order to save your work ***(Click)File > Save Project*** as shown in the image below.



Step 5: Solution

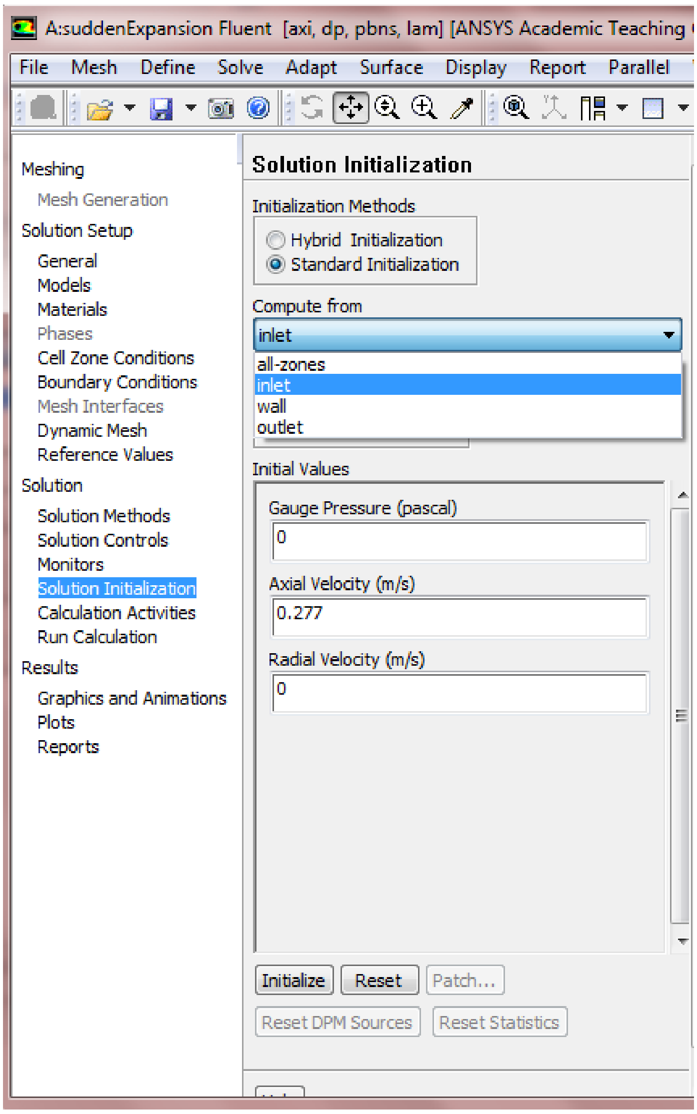
Second Order Scheme

A second-order discretization scheme will be used to approximate the solution. In order to implement the second order scheme click on ***Solution Methods*** then click on ***Momentum*** and select ***Second Order Upwind*** as shown in the image below.



Set Initial Guess

Here, the flow field will be initialized to the values at the inlet. In order to carry out the initialization click on ***Solution Initialization, Standard Initialization***, and click on ***Compute from*** and select ***inlet*** as shown below.

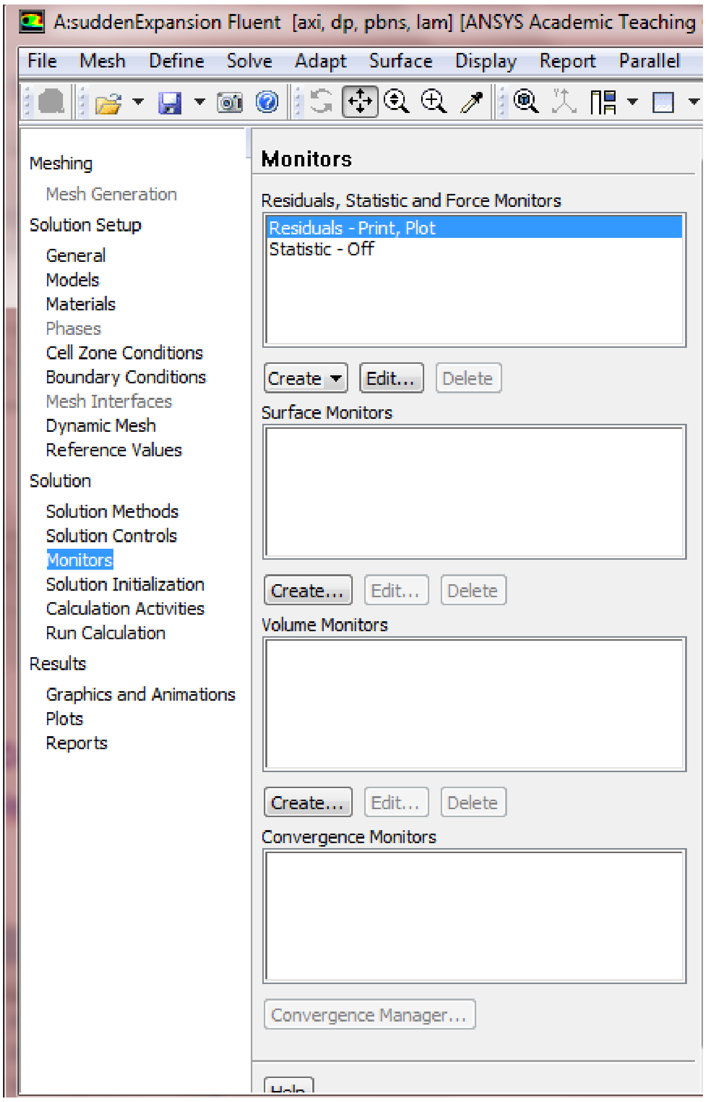


Then, click the ***Initialize*** button, 

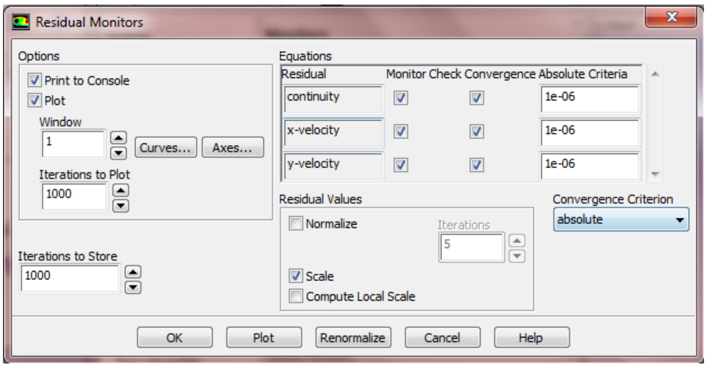
. This completes the initialization.

Set Convergence Criteria

FLUENT reports a residual for each governing equation being solved. The residual is a measure of how well the current solution satisfies the discrete form of each governing equation. We'll iterate the solution until the residual for each equation falls below 1e-6. In order to specify the residual criteria ***(Click) Monitors > Residuals > Edit...***, as shown in the image below.



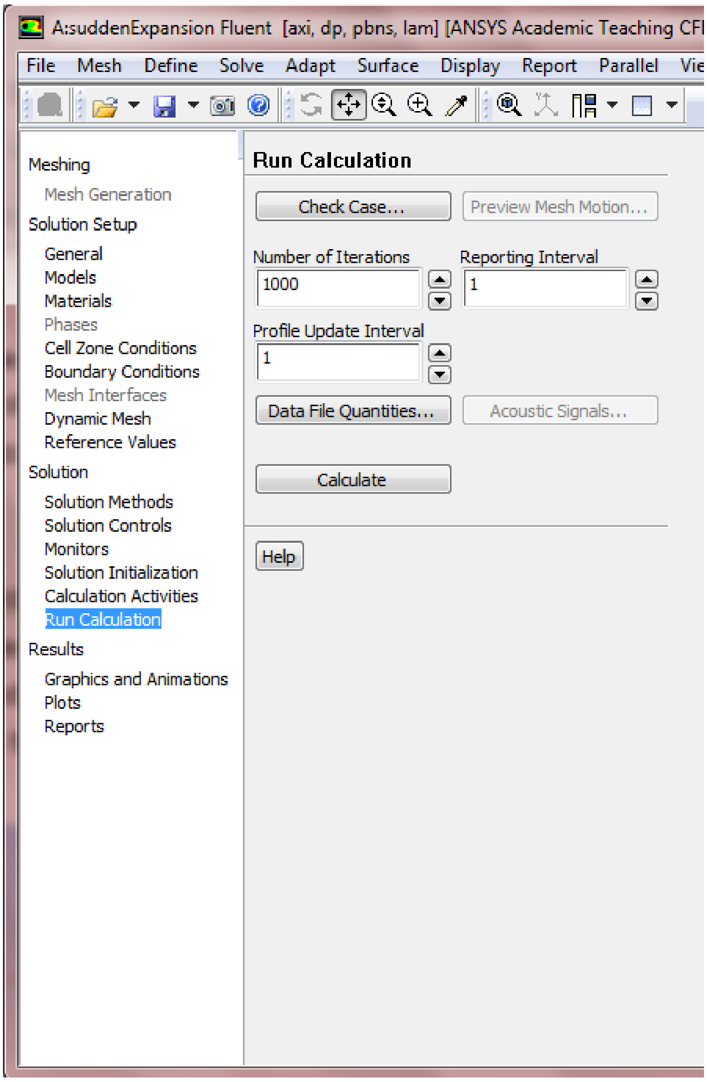
Next, change the residual under ***Convergence Criterion*** for ***continuity***, ***x-velocity***, and ***y-velocity***, all to 1e-6, as can be seen below.



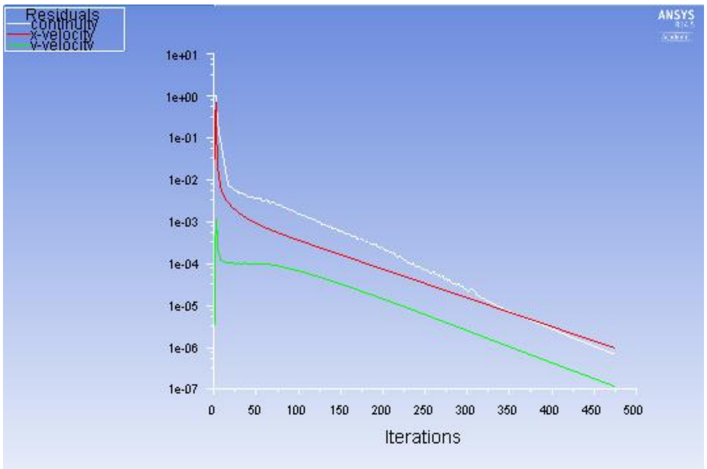
Lastly, click ***OK*** to close the ***Residual Monitors*** menu.

Execute Calculation

Prior, to running the calculation the maximum number of iterations must be set. To specify the maximum number of iterations click on ***Run Calculation*** then set the ***Number of Iterations*** to 1000, as shown in the image below.



As a safeguard save the project now. Now, click on ***Calculate*** two times in order to run the calculation. The residuals for each iteration are printed out as well as plotted in the graphics window as they are calculated. After running the calculation, you should obtain the following residual plot.



The residuals fall below the specified convergence criterion of 1e-6 in about 473 iterations, as shown below. Actual number of convergence steps may vary slightly.

```
    472  7.1711e-07  1.0157e-06  1.2395e-07  0:00:38  528
!   473 solution is converged
    473  7.0347e-07  9.9975e-07  1.2188e-07  0:00:30  527
```

At this point, save the project once again.

Step 6: Results

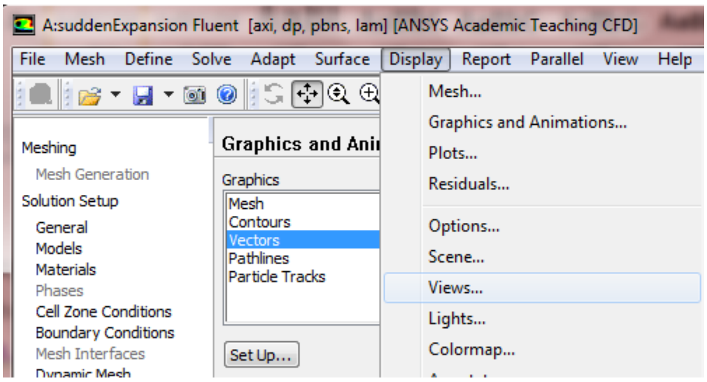
The results steps shown below are for the CFD-Post postprocessor that is included in ANSYS Workbench.

Velocity Vectors

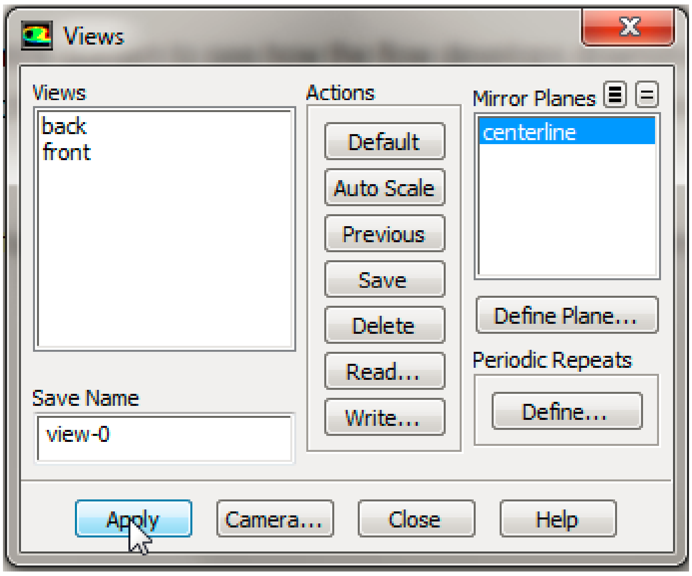
One can plot vectors in the entire domain, or on selected surfaces. Let us plot the velocity vectors for the entire domain to see how the flow develops downstream from the inlet and redevelops from the expansion entrance. First, click on ***Graphics & Animations***. Next, double click on ***Vectors*** which is located under ***Graphics***. Then, click on ***Display***. Zoom into the region near the inlet. The length and color of the arrows represent the velocity magnitude. The vector display is more intelligible if one makes the arrows shorter as follows:

Change ***Scale*** to `0.4` in the ***Vectors*** menu and click ***Display***.

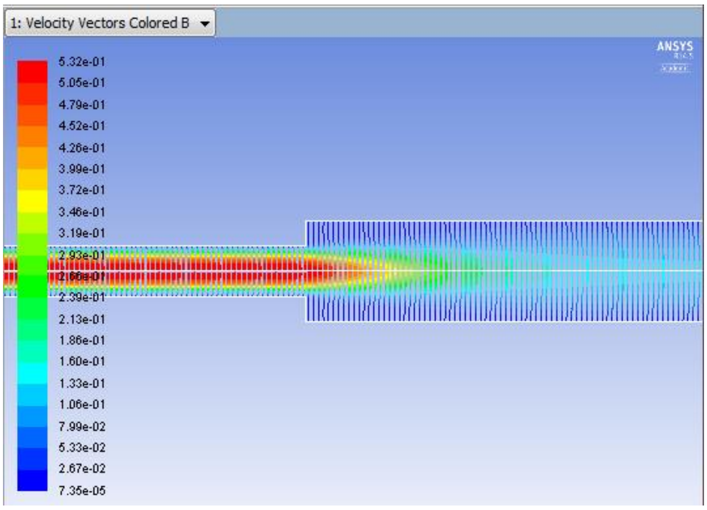
The laminar pipe flow was modeled asymmetrically; however, the plot can be reflected about the axial axis to get an expanded sectional view. In order to carry this out ***(Click) Display > Views...*** as shown below.



Under ***Mirror Planes***, only the ***axis (or centerline)*** surface is listed since that is the only symmetry boundary in the present case. Select ***axis (or centerline)*** and click ***Apply***, as shown below.



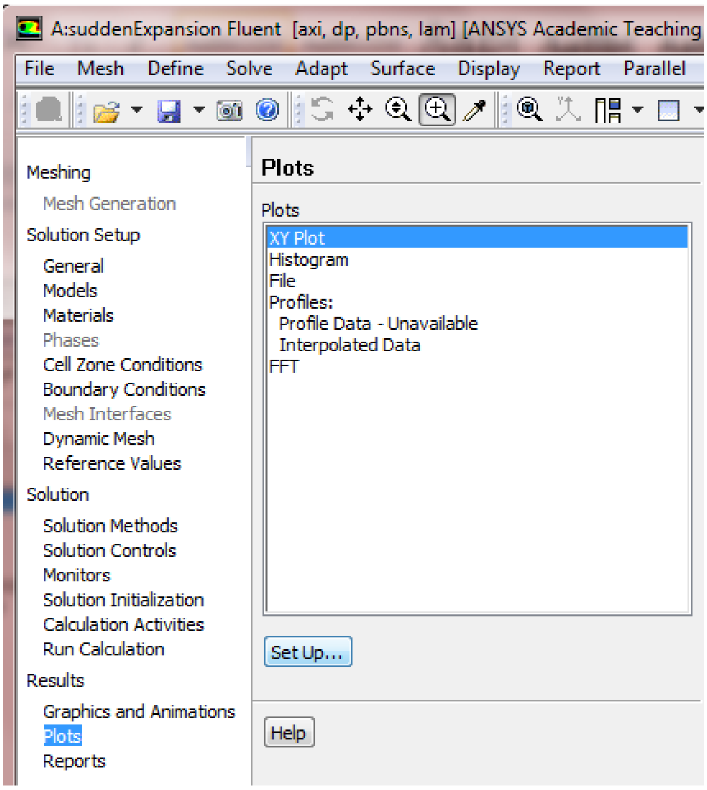
Then click ***Close*** to exit the ***Views*** menu. Your vector field should have been reflected across the axially axis as, shown below.



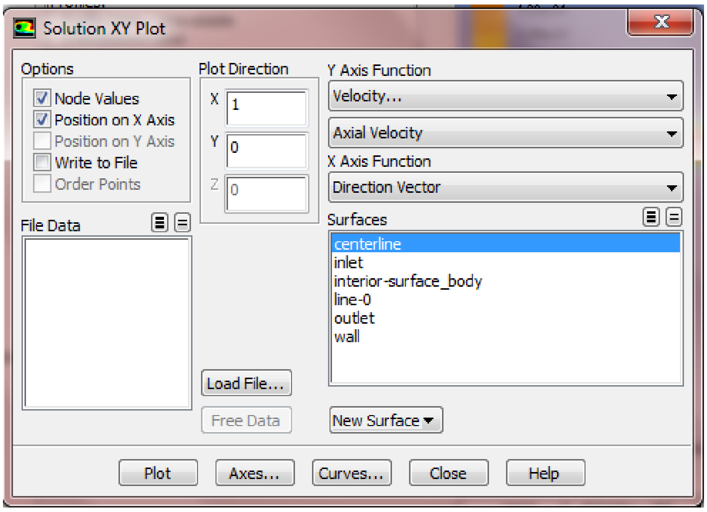
The velocity vectors provide a picture of how the flow develops downstream from the inlet and redevelops from the expansion entrance.

Centerline Velocity

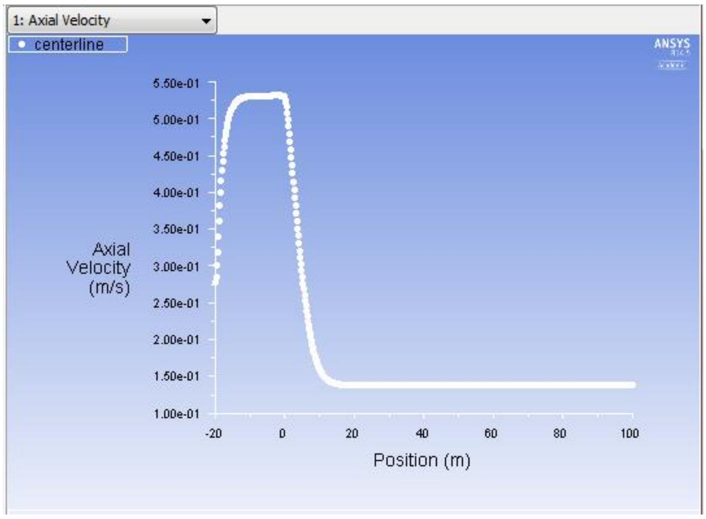
Here, we'll plot the variation of the axial velocity along the centerline. In order to start the process ***(Click) Results > Plots > XY Plot... > Set Up..*** as shown below



In the ***Solution XY Plot*** menu make sure that ***Position on X Axis*** is selected, and ***X*** is set to `1` and ***Y*** is set to `0`. This tells FLUENT to plot the x-coordinate value on the abscissa of the graph. Next, select ***Velocity...*** for the first box underneath ***Y Axis Function*** and select ***Axial Velocity*** for the second box. Please note that ***X Axis Function*** and ***Y Axis Function*** describe the *x* and *y* axes of the *graph*, which should not be confused with the *x* and *y* directions of the pipe. Finally, select ***centerline*** under ***Surfaces*** since we are plotting the axial velocity along the centerline. This finishes setting up the plotting parameters. Your ***Solution XY Plot*** should look exactly the same as the following image.



Now, click ***Plot***. The plot of the axial velocity as a function of distance along the centerline now appears.



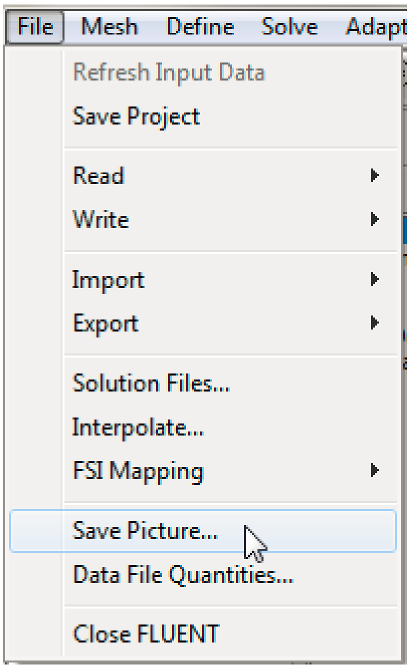
In the graph that comes up, we can see that the velocity reaches a constant value (about 5.25e-1 m/s) beyond a certain distance from the inlet. This is the fully-developed flow region in the small pipe. When the flow pass the expansion entrance at x=0 m, the velocity will decrease due to the sudden expansion of the pipe. After another development, the velocity will reach a constant value (about 1.5e-1m/s) again.

Saving the Plot

In this section, we will save the data from the plot and a picture of the plot. The data from the plot will be saved first. In order to save the plot data open the ***Solution XY Plot*** menu and then select ***Write to File***, which is located under

Options. The ***Plot*** button should have changed to ***Write...***. Click on ***Write...***, and then enter `vel.xy` as the *XY* File Name. Next, click ***OK***. Check that this file has been created in your FLUENT working directory. Lastly, close the ***Solution XY Plot*** menu.

At this point, we'll save a picture of the plot. First click on ***File*** then click ***Save Picture***, as shown below.



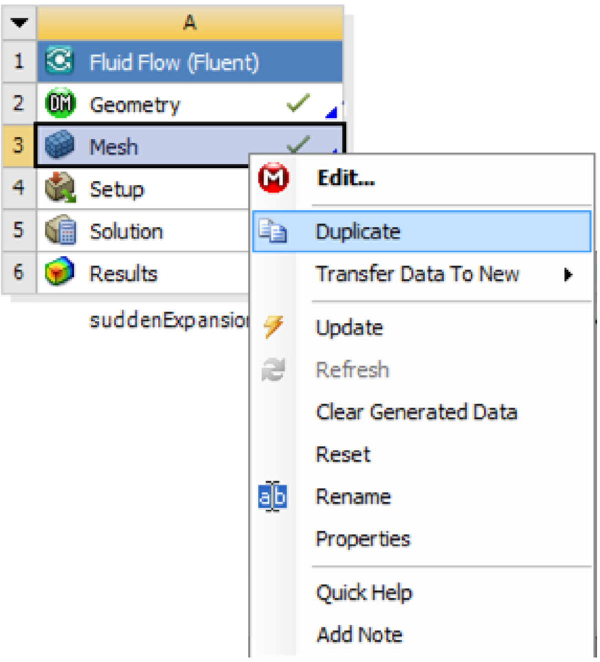
Under ***Format***, choose one of the following three options: ***EPS***, ***TIFF***, or ***JPG***. After selecting your desired image format and associated options, click on ***Save...*** Enter `vel.eps`, `vel.tif`, or `vel.jpg` depending on your format choice and click ***OK***. Verify that the image file has been created in your working directory. You can now copy this file onto a disk or print it out for your records.

Step 7: Verification & Validation

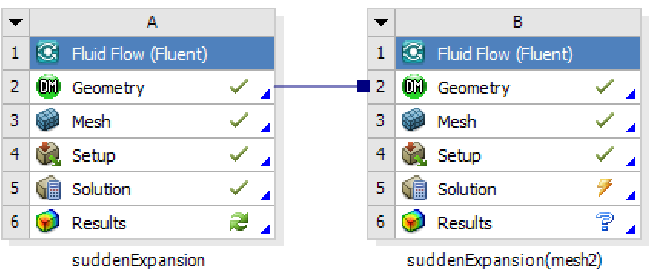
It is ***very important*** that you take the time to check the validity of your solution. Based on the velocity profile along the centerline, it can be seen that the max axis velocity in the small pipe is about 5.25e-1 m/s. However, in a full-developed laminar pipe flow, the analytical max axis velocity 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

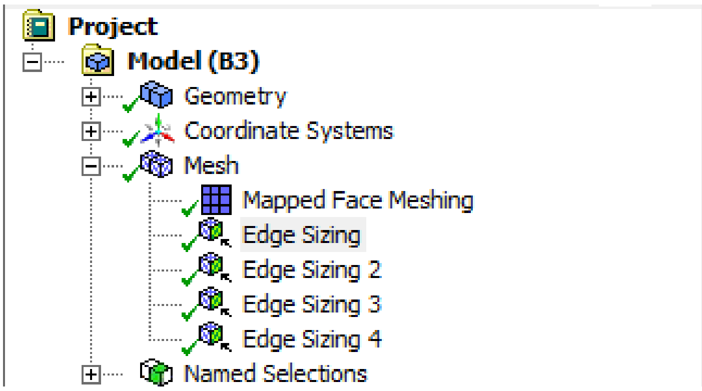
Let's repeat the solution on a finer mesh. For the finer mesh, the numbers of radial divisions will be twice as many as those used before. In the ***Workbench Project Page*** right click on ***Mesh*** then click ***Duplicate*** as shown below.



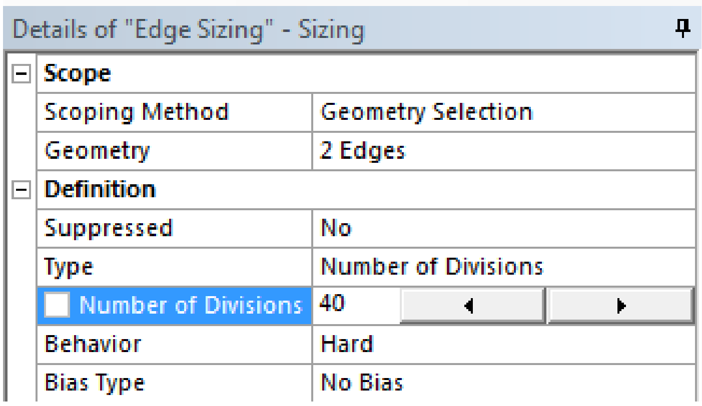
Rename the duplicate project to ***suddenExpansion (mesh2)***. You should have the following two projects in your ***Workbench Project Page***.



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.



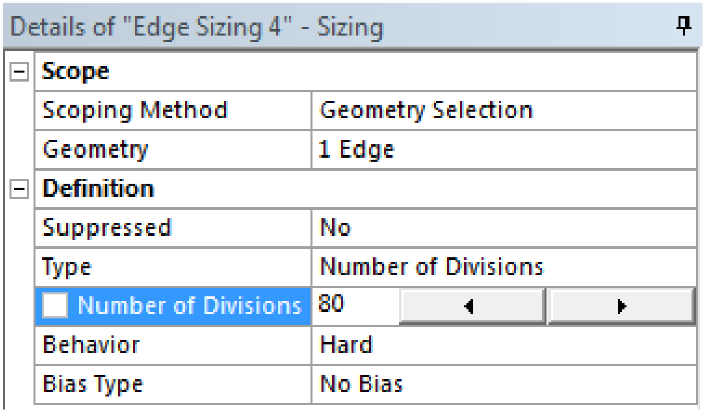
Here we will increase the numbers of cells in the radial direction. You may have different names of sizings. In my case, the "Edge Sizing" corresponds to the AB and EF edges, and “Edge Sizing 4” corresponds to the CD edge. Double the ***Number of Divisions*** of “Edge Sizing”



Details of "Edge Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	2 Edges
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	40
Behavior	Hard
Bias Type	No Bias

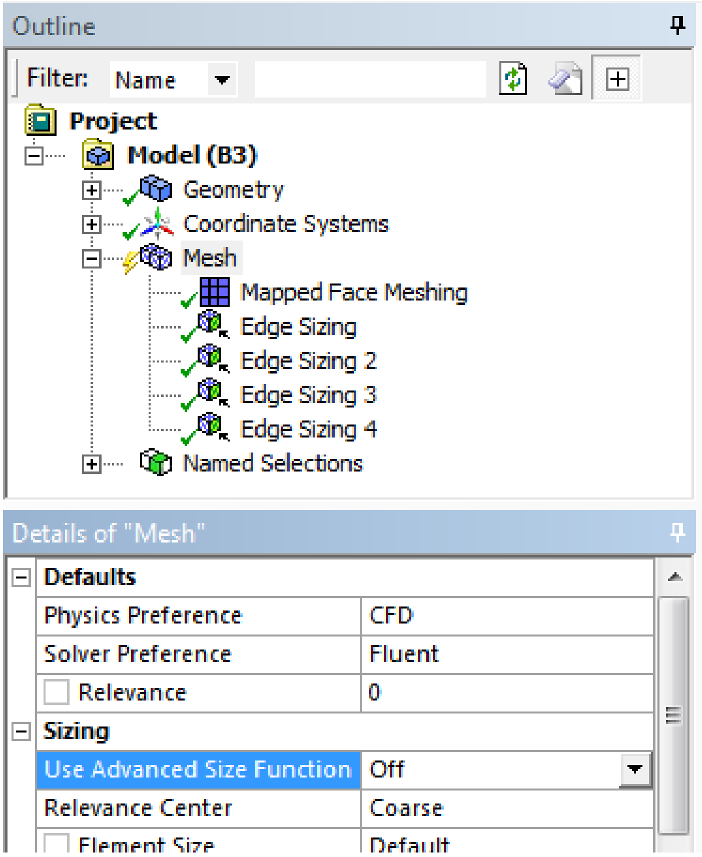
and “Edge Sizing 4”



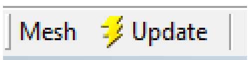
Details of "Edge Sizing 4" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	80
Behavior	Hard
Bias Type	No Bias

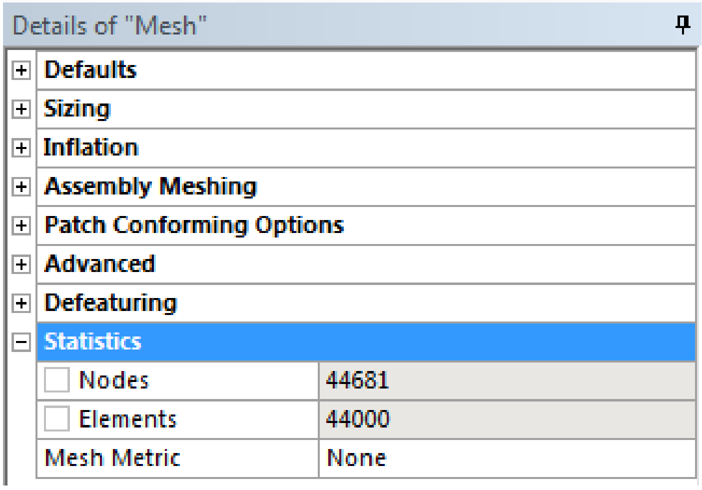
Sometimes, you need to turn-off "**Advanced Size Function**" under "**Details of Mesh**" to get the mesher to accept the modified settings. That way the Advanced Size Function feature will not over-ride your settings (this feature is useful for meshing complex geometries). Click ***Mesh*** in the tree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



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

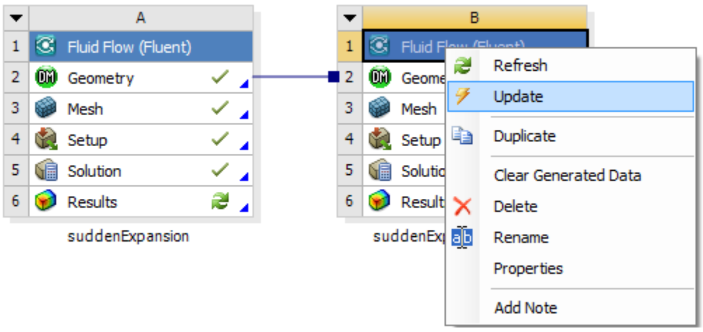


The mesh should now have 44000 elements. A quick glance of the mesh statistics reveals that there are indeed 44000 elements.



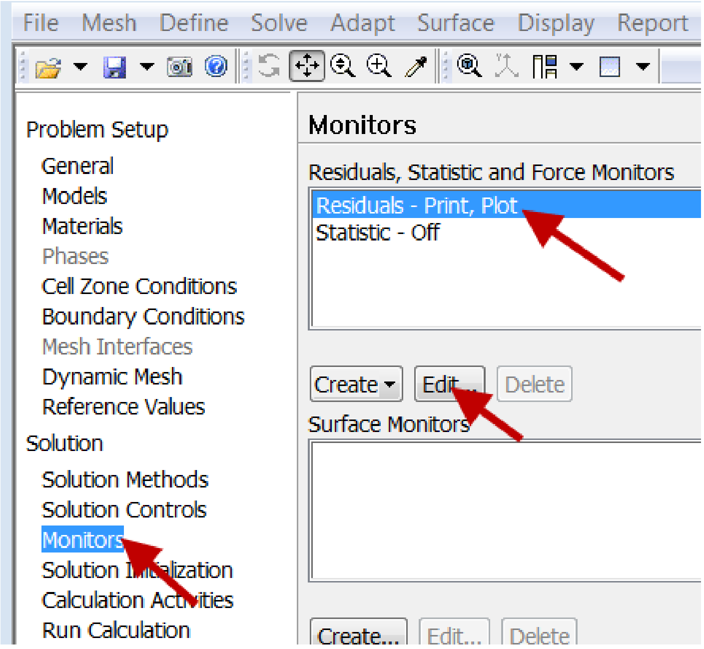
Compute the Solution

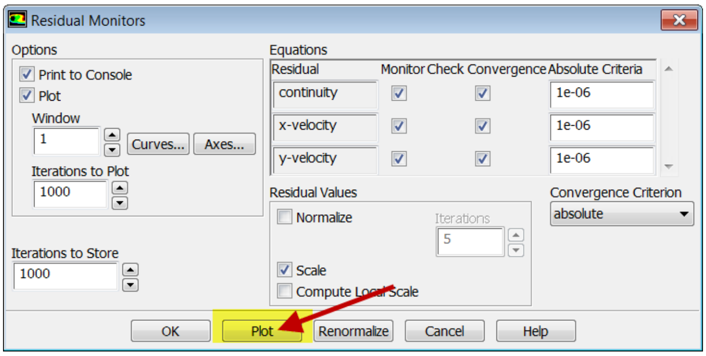
Close the ANSYS Mesher to go back to the ***Workbench Project Page***. Under ***suddenExpansion (mesh2)***, right click on ***Fluid Flow (FLUENT)*** and click on ***Update***, as shown below.



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.

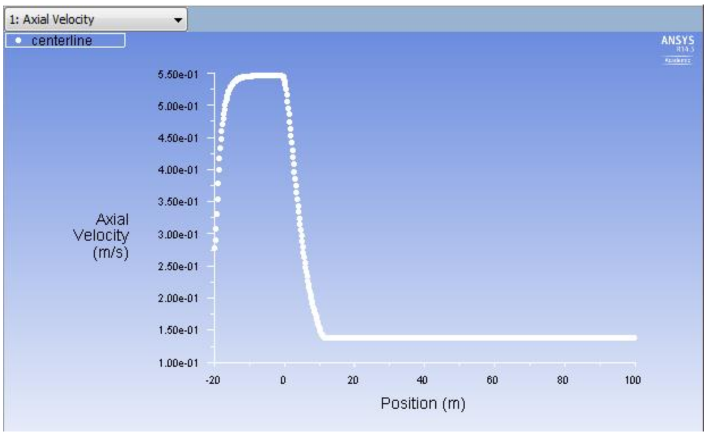




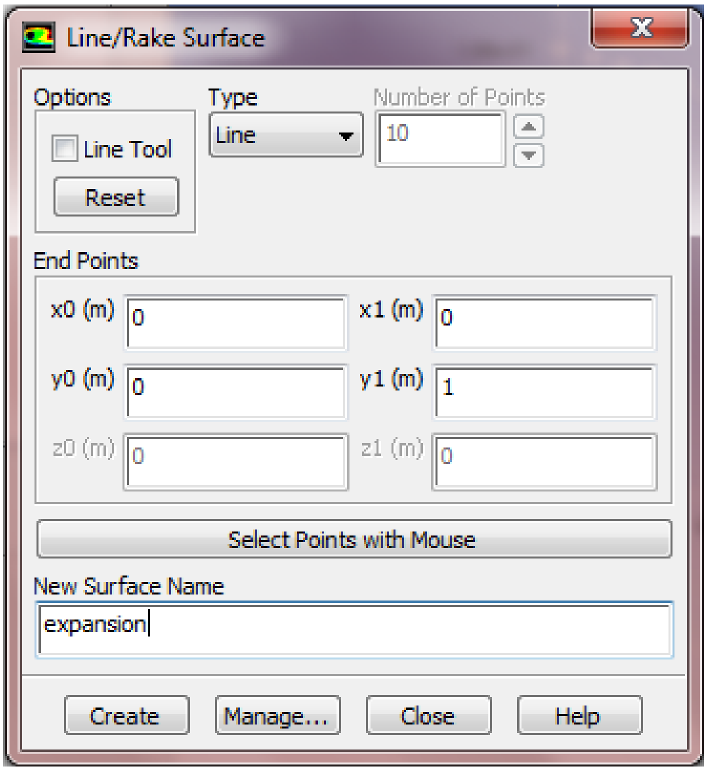
If the solution hasn't converged, one needs to run more iterations by selecting ***Run Calculation***. You may want to increase the number of iterations. Ensure that you have a converged solution and save the project. Then, if you double-click on ***Results*** for mesh2 in the project page, you'll see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

Velocity profile

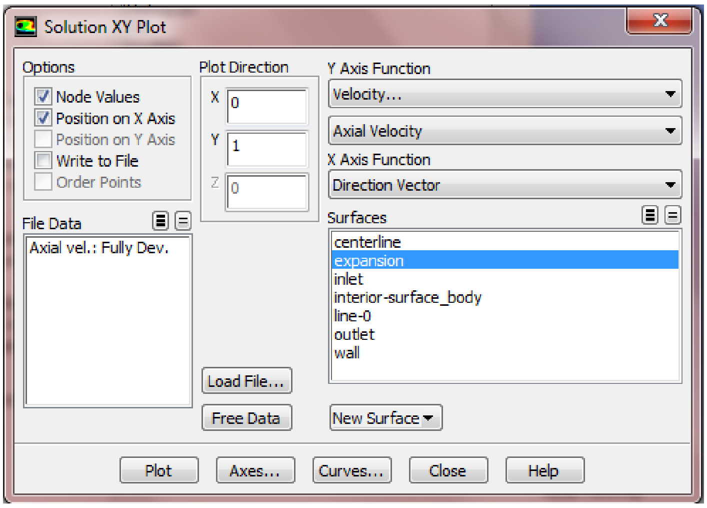
The plot below shows the profile of the axial velocity along the centerline (y = 0 m) with the refined mesh. It can be seen that the max velocity in the small pipe is about 5.50e-1 m/s, which is close to the analytical solution (5.54e-1 m/s) and better than that of the previous mesh.



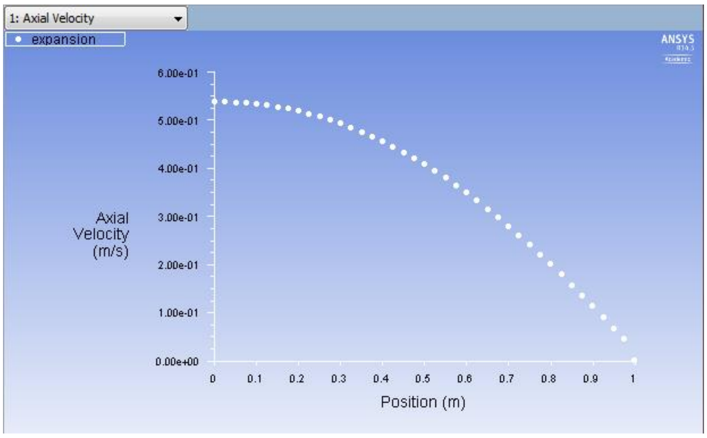
Here we will create a straight line for the expansion entrance, which is from (x0, y0) = (0, 0) to (x1, y1) = (0, 1). Select ***Line*** under ***Surface***. Enter ***x0*** = 0, ***y0*** = 0, ***x1*** = 0, ***y1*** = 1. Enter expansion under ***New Surface Name***. Click ***Create***.



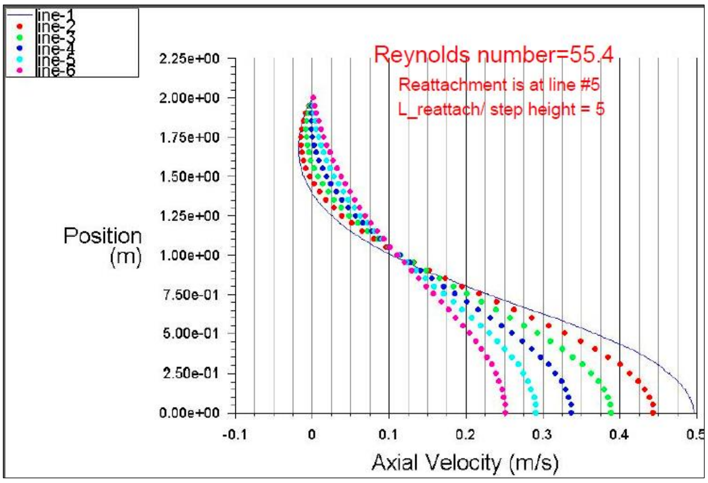
Then, plot the velocity profile along the ***expansion*** line. In the ***Solution XY Plot*** menu make sure that ***Position on X Axis*** is selected, and ***X*** is set to `0` and ***Y*** is set to `1`.



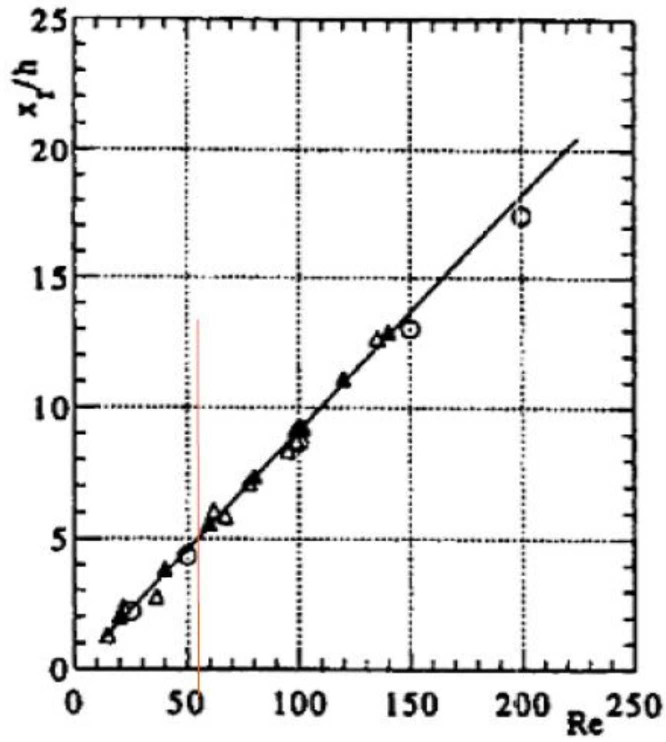
Then, click ***Plot*** and you should obtain the following output.



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.

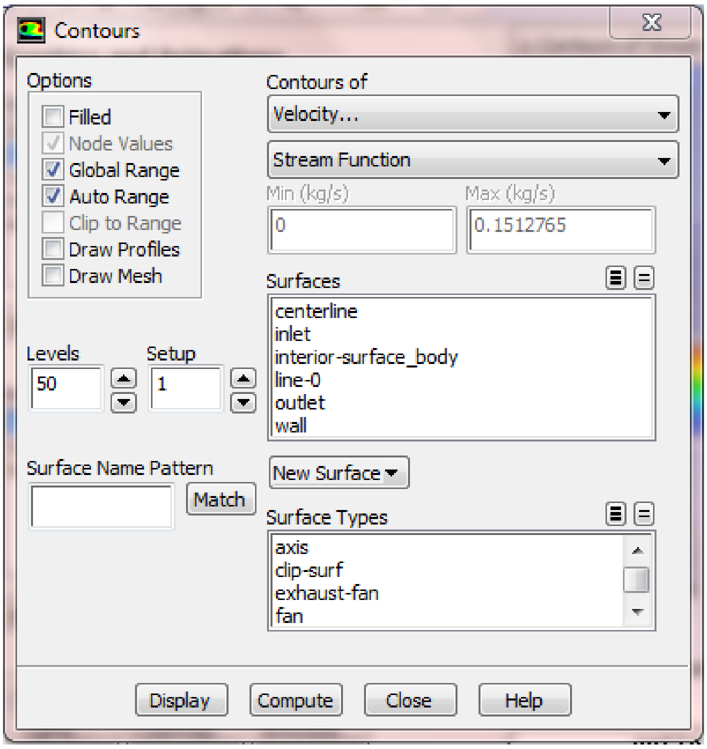


As you can see from the velocity profiles, the end of the recirculation region is at ***x*** = 5 m. In other words, the recirculation length for *Re* = 55.4 is about 5 times of the inlet radius, which agree well with the experimental data (*Hammad et al., Experiments in Fluids, 1999, 26:266-271*) shown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

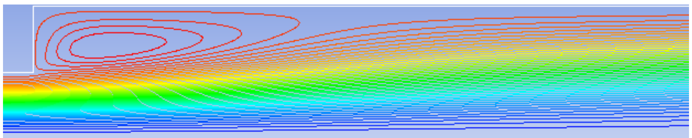


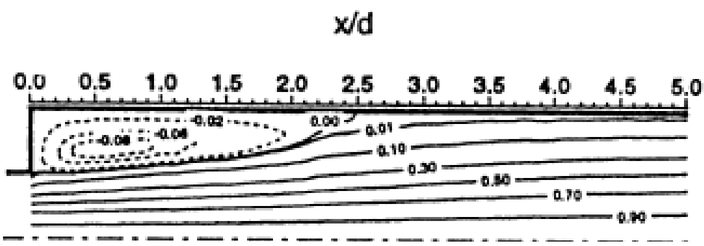
Streamline

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 blow.



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.