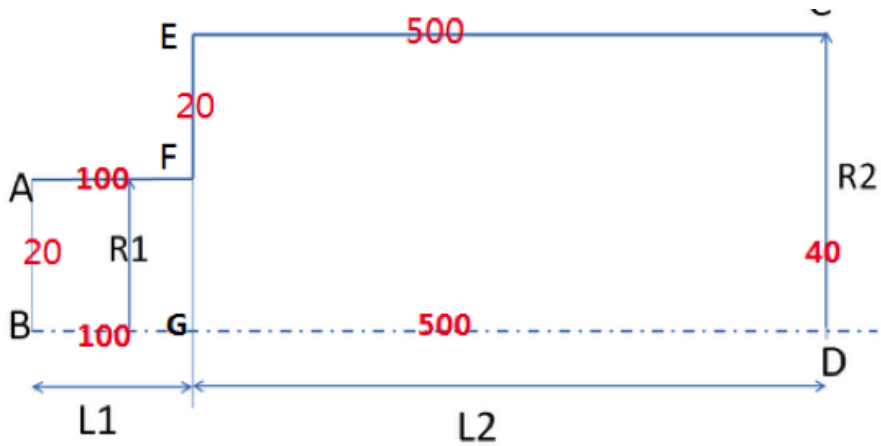
Sudden Expansion - Mesh

Authors: Yong Wang & Said Elghobashi, UC Irvine

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:

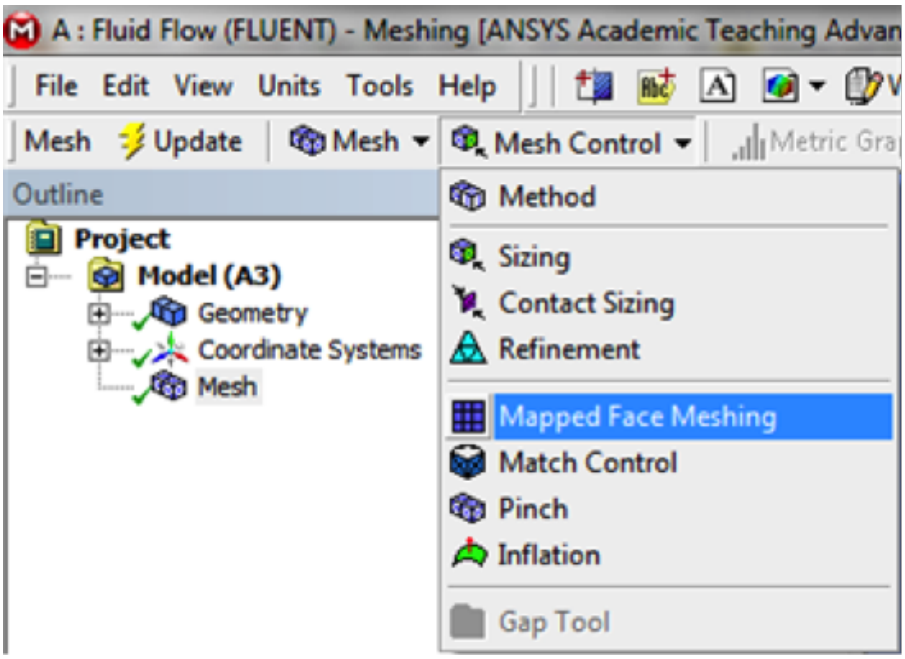
Launch Mesher

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

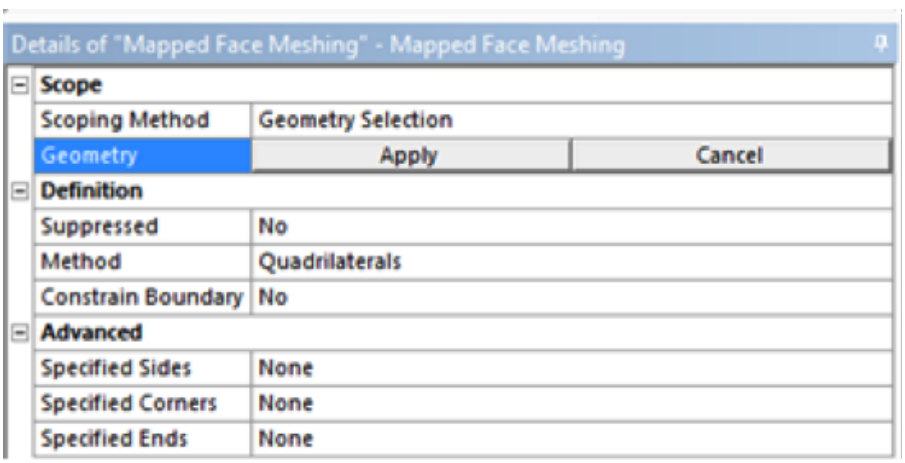
You may want to generate a preliminary mesh by right clicking on ***Mesh*** (upper window) and then "***Generate Mesh***".

Mapped Face Meshing

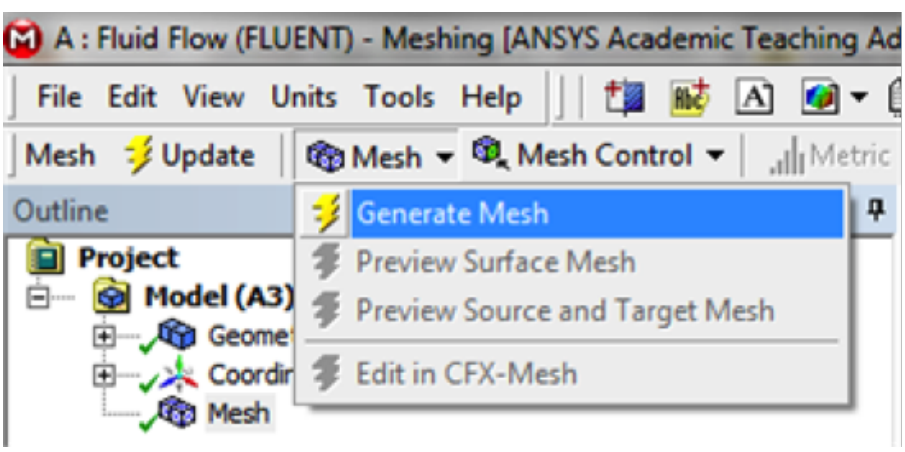
Here we are interested in creating a grid style of mesh that can be mapped to a rectangular domain. This meshing style is called ***Mapped Face Meshing***. In order to incorporate this meshing style ***(Click) Mesh Control > Mapped Face Meshing*** as can be seen below.



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.

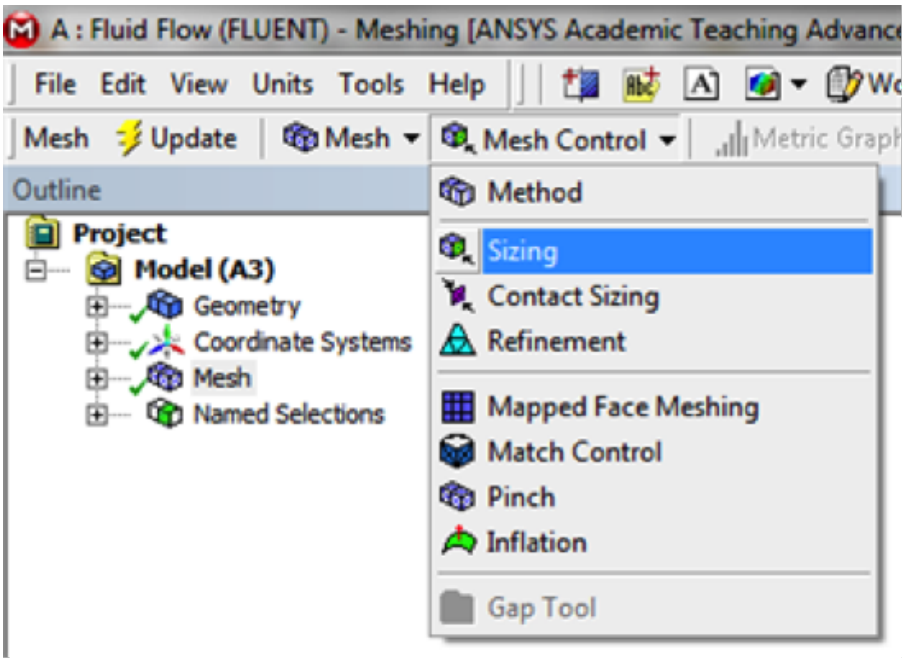


After clicking ***Apply***, you should see "***Geometry: 1Face***". Now, ***(Click) Mesh > Generate Mesh*** as can be seen below, and generate a rough mesh.

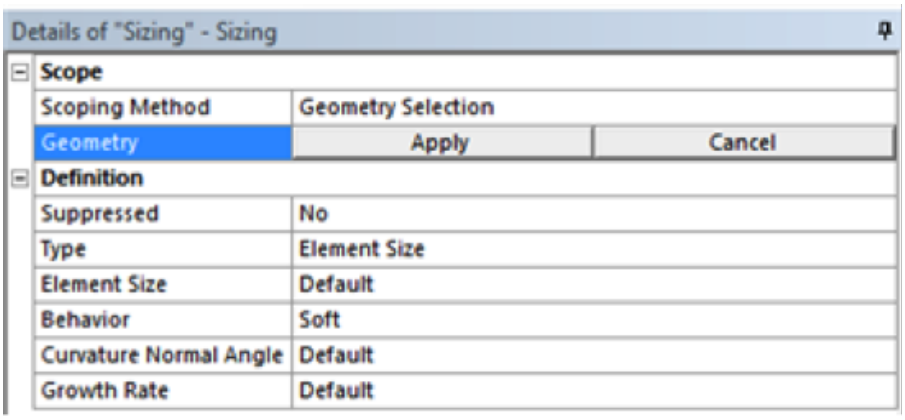


Edge Sizing

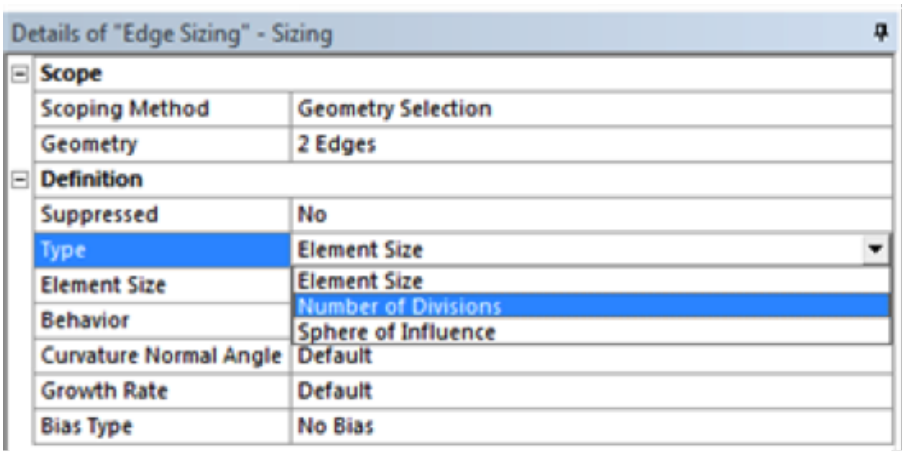
The desired mesh has specific number of divisions along the radial and the axial direction. In order to obtain the specified number of divisions ***Edge Sizing*** must be used. The divisions along the axial direction will be specified first. Now, an ***Edge Sizing*** needs to be inserted. First, ***(Click)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



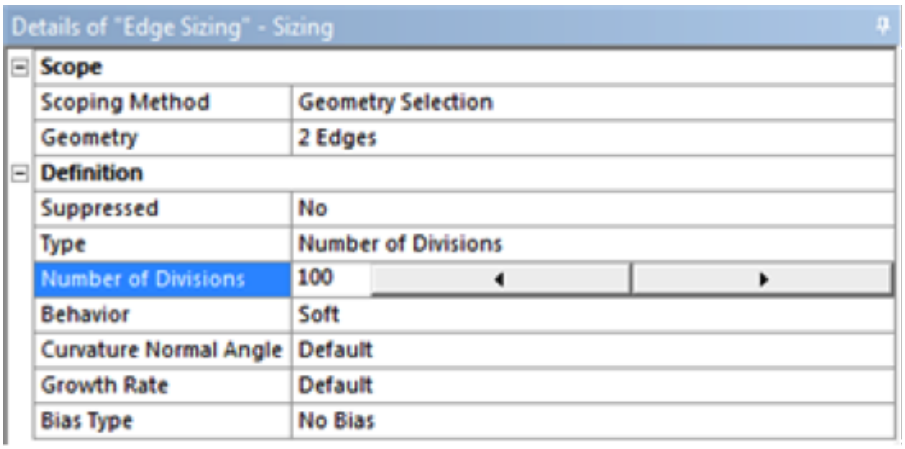
Now, the geometry and the number of divisions need to be specified. First ***(Click) Edge Selection Filter***,  , in the upper toolbar. Hold down the "Ctrl" button and then click the edges ***AF*** and ***BG***. Both sides should highlight green. Next, hit ***Apply*** under the ***Details of Sizing***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



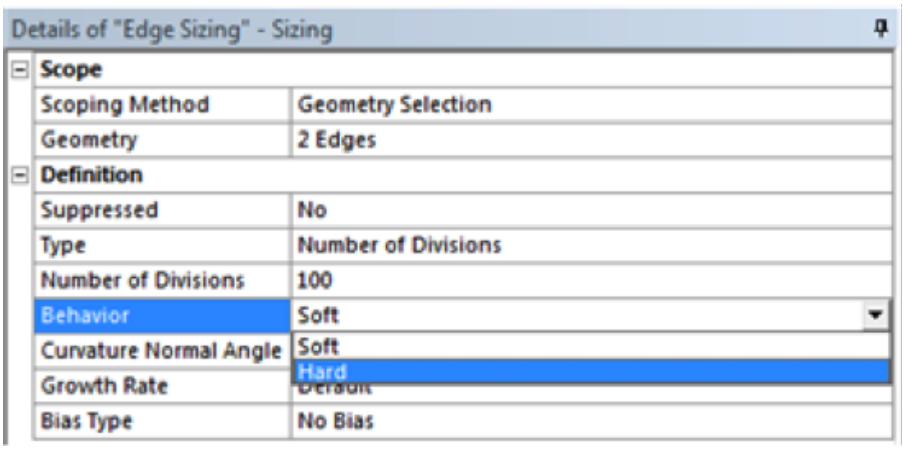
Now, change ***Type*** to ***Number of Divisions*** as shown in the image below.



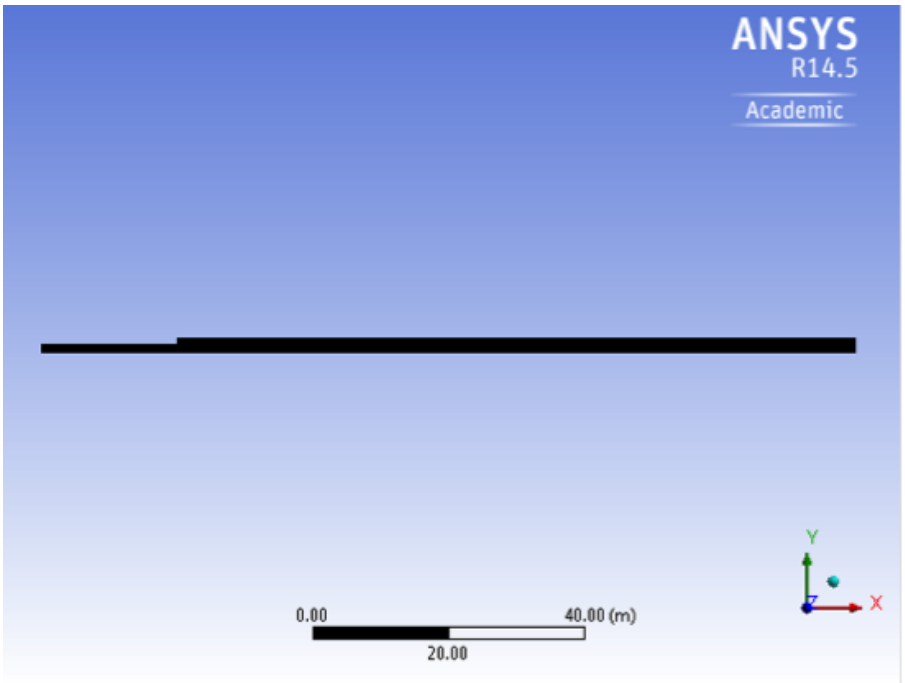
Then, set ***Number of Divisions*** to 100 as shown below.



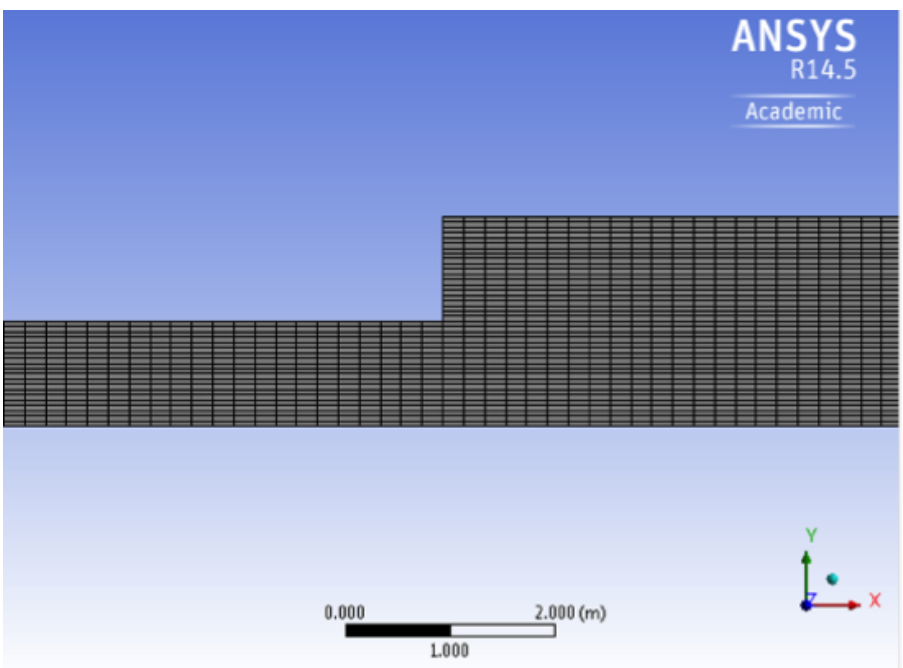
Change ***Behavior*** to ***Hard*** for both ***Edge Sizing's***.



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.



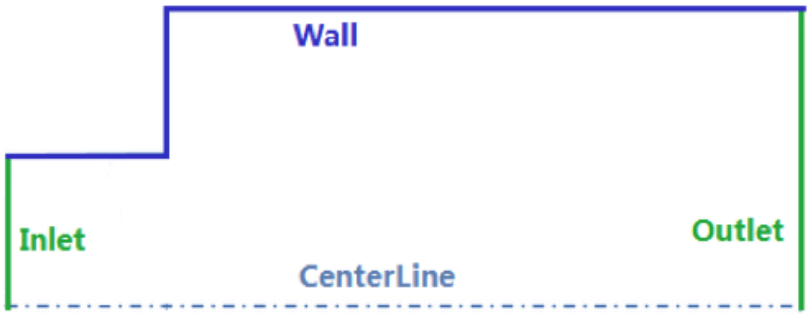
Detail around the expansion entrance FG is presented blow.



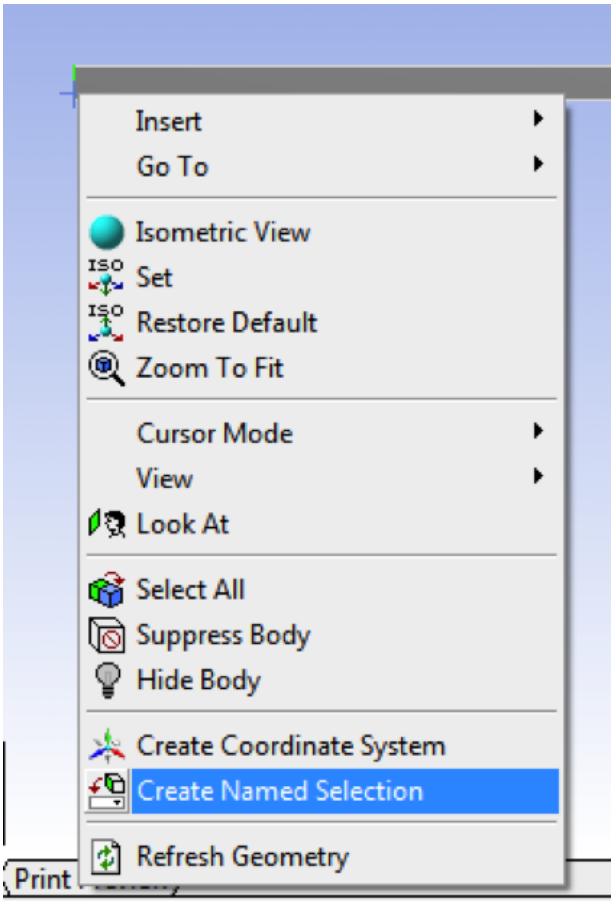
Mesh statistics can be found by clicking on ***Mesh*** in the tree and then by expanding ***Statistics*** under the ***Details of Mesh*** table. It can be seen that there are 22000 elements in the mesh above.

Create Named Selections

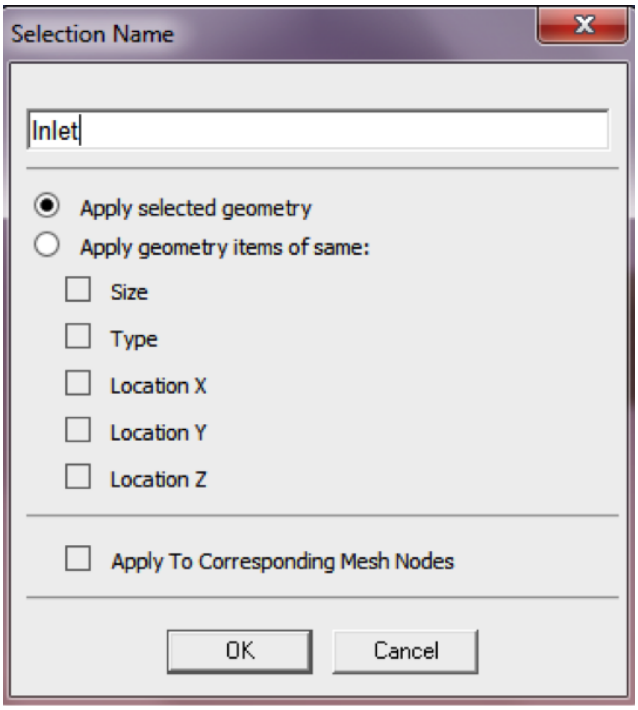
Here, the edges of the geometry will be given names so one can assign boundary conditions in Fluent in later steps. The left side of the pipe will be called "Inlet" and the right side will be called "Outlet". The bottom side will be called "CenterLine" and the other edges are called "Wall", as shown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

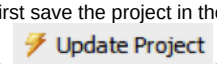


Select the left edge and right click and select ***Create Named Selection***. Enter Inlet and click ***OK***, as shown below.



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*** butto n, 

.

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