

Sudden Expansion - Physics Setup

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](#)

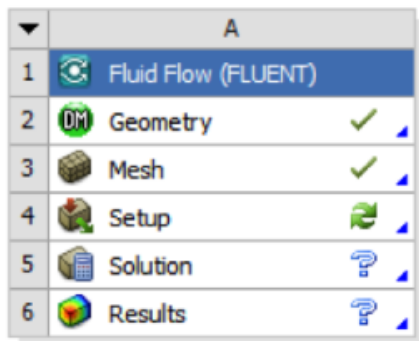
[7. Verification & Validation](#)

[Exercises](#)

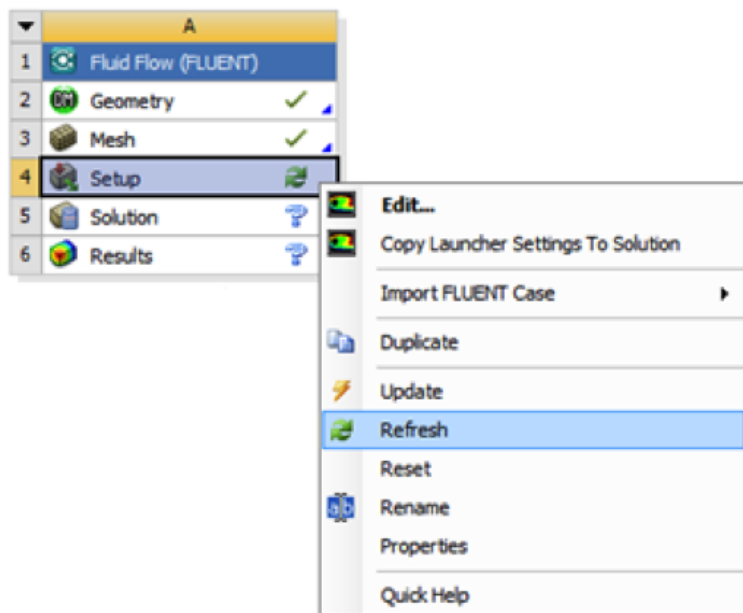
[Comments](#)

Physics Setup

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**.



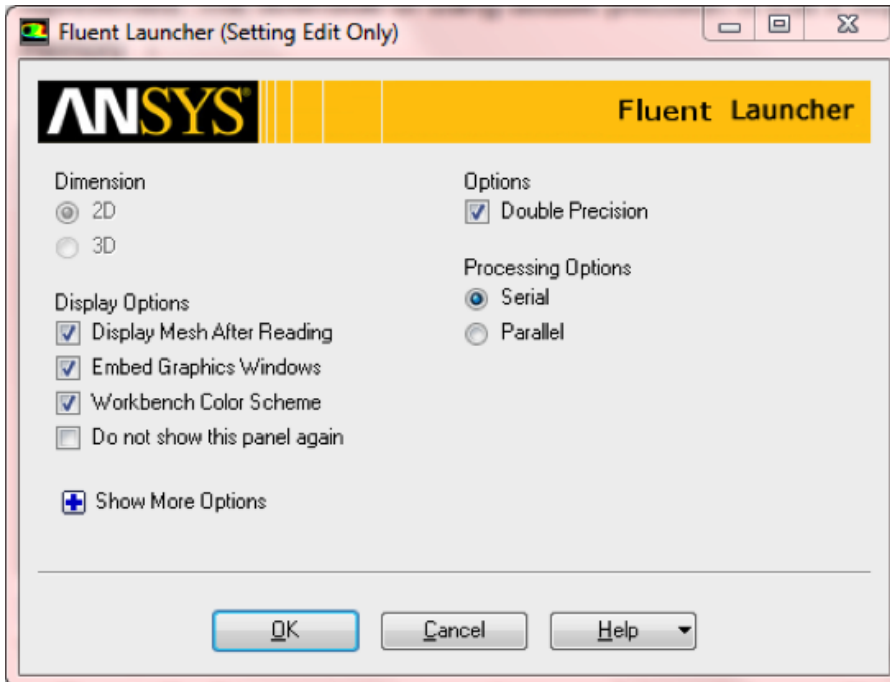
Next, the mesh and geometry data need to be read into FLUENT. To read in the data (**Right Click**) **Setup** > **Refresh** in the **Workbench Project Page** as shown in the image below.



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

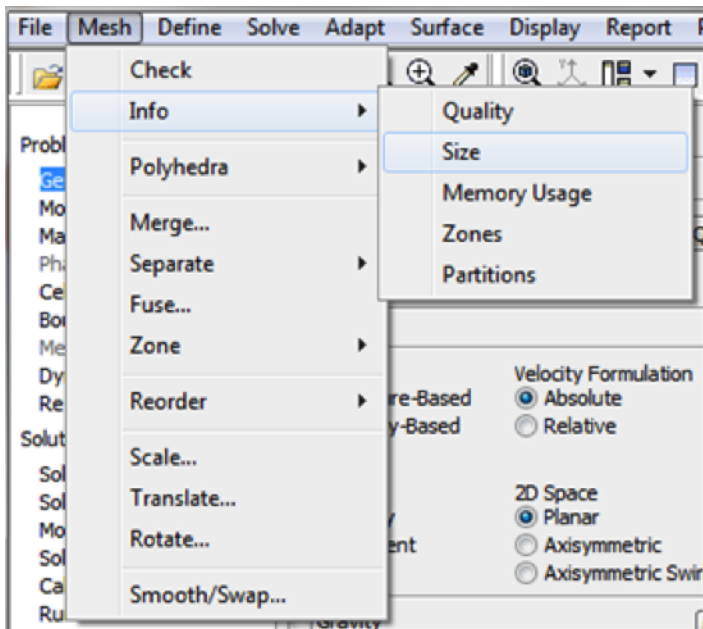
Double click on **Setup** in the **Workbench Project Page** which will bring up the **FLUENT Launcher**. When the **FLUENT Launcher** appears change the options to "Double Precision", and then click **OK** as shown below. The Double Precision option is used to select the double-precision solver. In the 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.



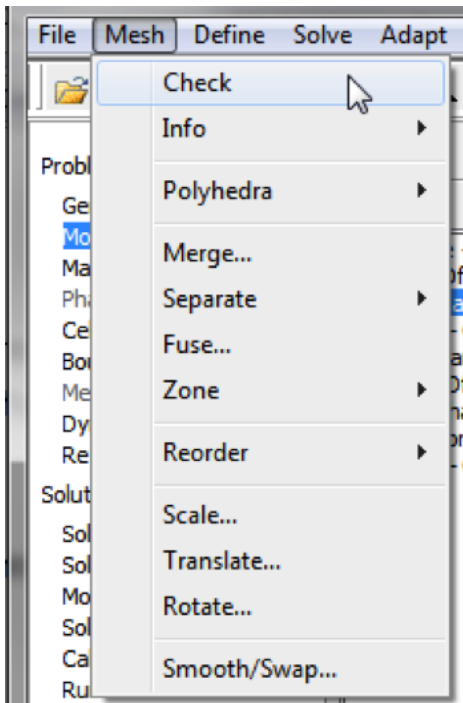
Then, you should obtain the following output in the **Command** pane.

Mesh Size

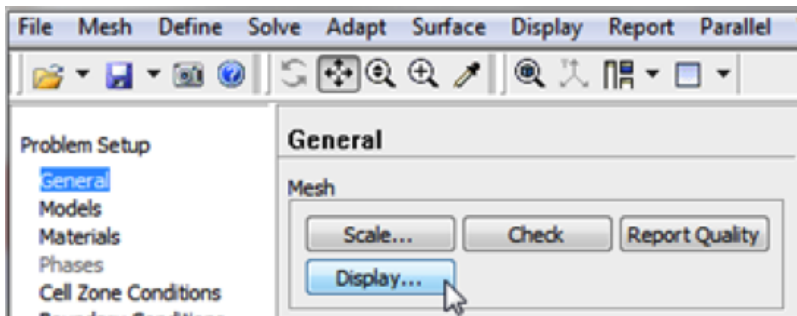
Level	Cells	Faces	Nodes	Partitions
0	22000	44640	22641	1

1 cell zone, 5 face zones.

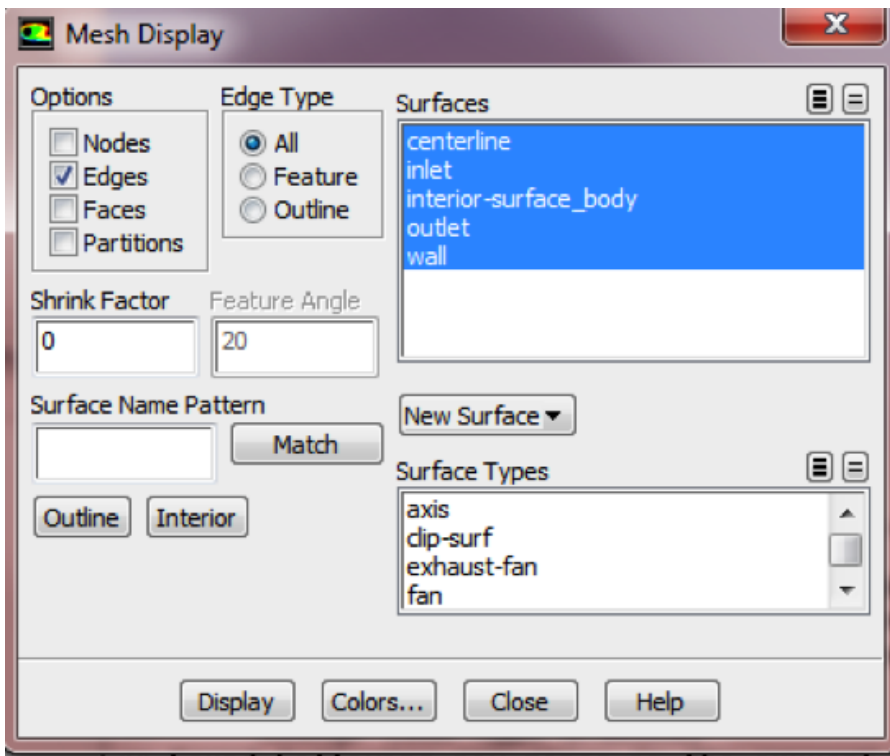
The mesh that was created earlier has 22000 elements. Note that in FLUENT elements are called cells. The output states that there are 22000 cells, which is a good sign. Next, FLUENT will be asked to check the mesh for errors. In order to carry out the mesh checking procedure (Click) **Mesh > Check**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



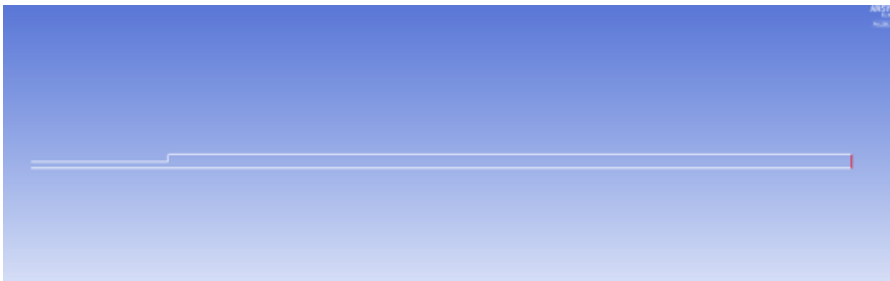
You should see no errors in the **Command** Pane. Now, that the mesh has been verified, the mesh display options will be discussed. In order to bring up the display options (**Click**) **General > Mesh > Display** as shown in the image below.





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.



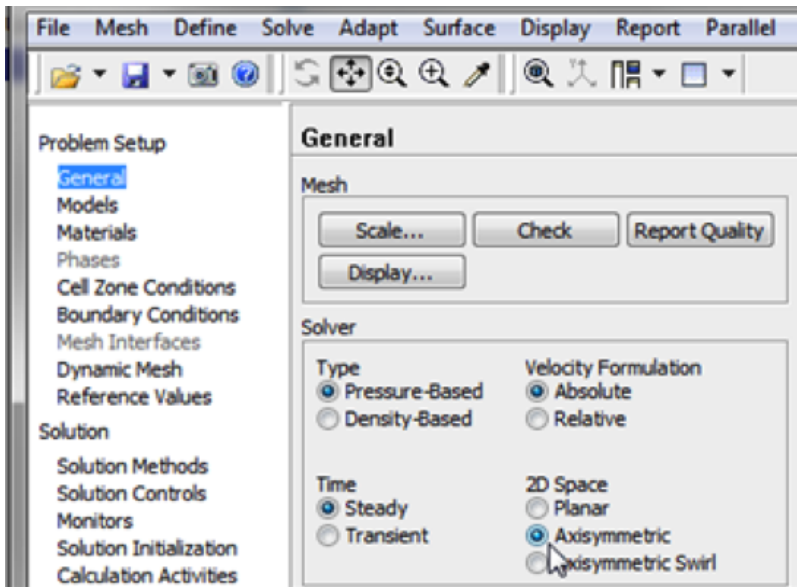
You should have all the surfaces shown in the above snapshot. Clicking on a surface name in the **Mesh Display** menu will toggle between select and unselect. Clicking **Display** will show all the currently selected surface entities in the graphics pane. Unselect all surfaces and then select each one 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.

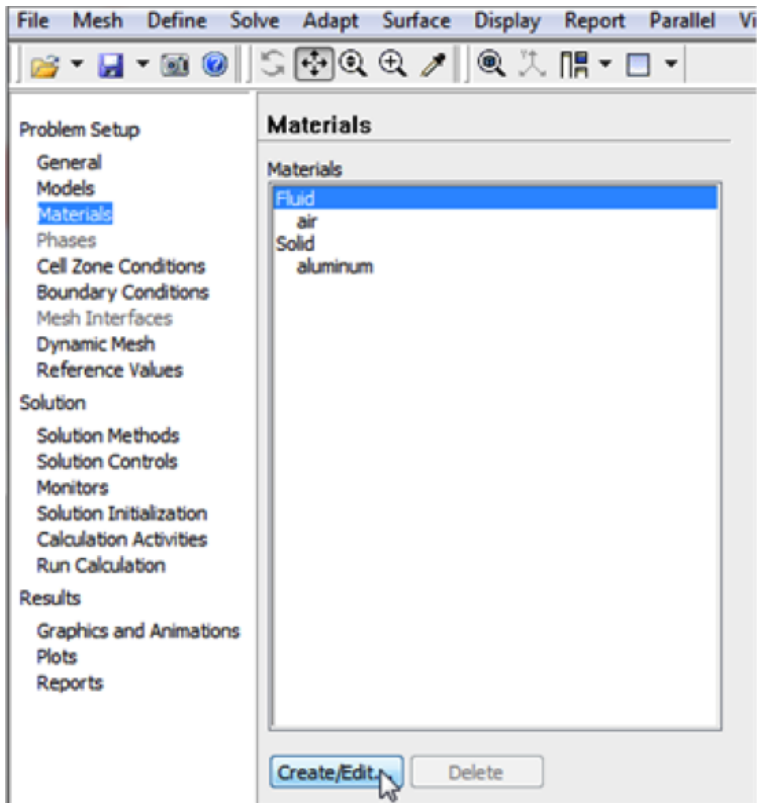


Next, the **Viscous Model** parameters will be specified. In order to open the Viscous Model Options **Models > Viscous - Laminar > Edit...** By default, the Viscous Model options are set to laminar, so no changes are needed. Click **Cancel** to exit the menu.

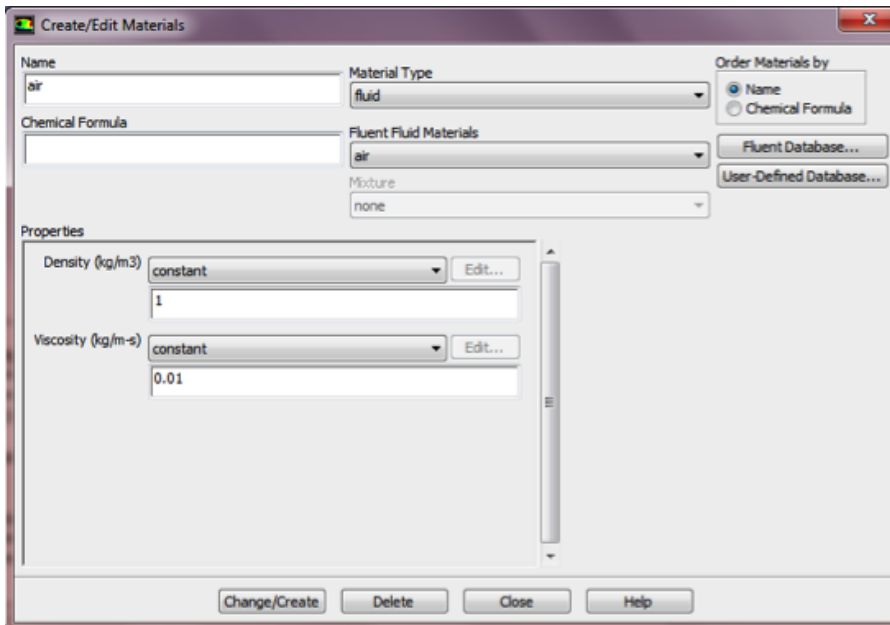
Now, the Energy Model parameters will be specified. In order to open the Energy Model Options **Models > Energy-Off > Edit...** For incompressible flow, the energy equation is decoupled from the continuity and momentum equations. We need to solve the energy equation only if we are interested in determining 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 $3.61\text{e-}2\text{ kg/(ms)}$ (constant) as shown in the image below.



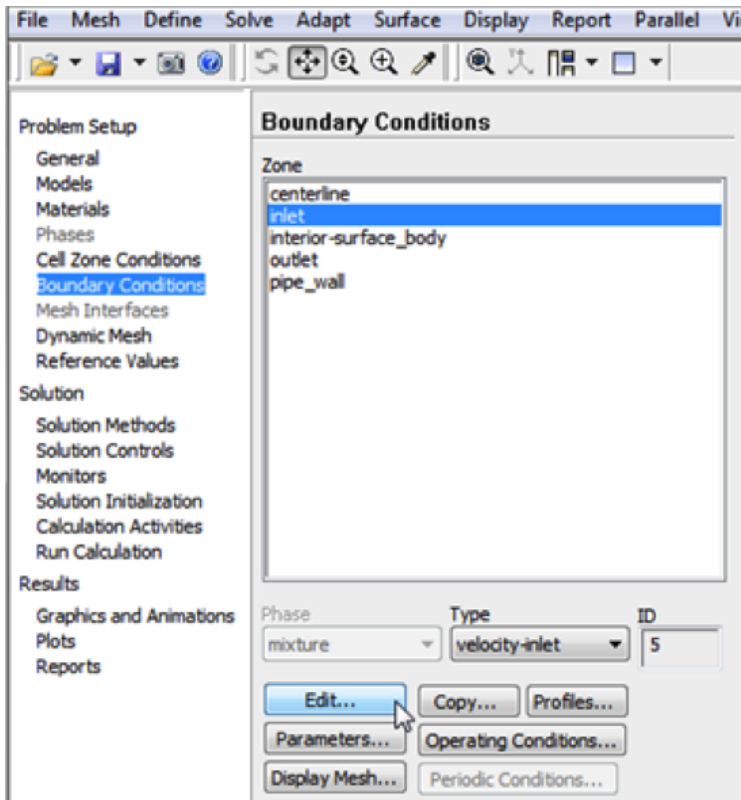
Click **Change/Create**. Close the window.

Define Boundary Conditions

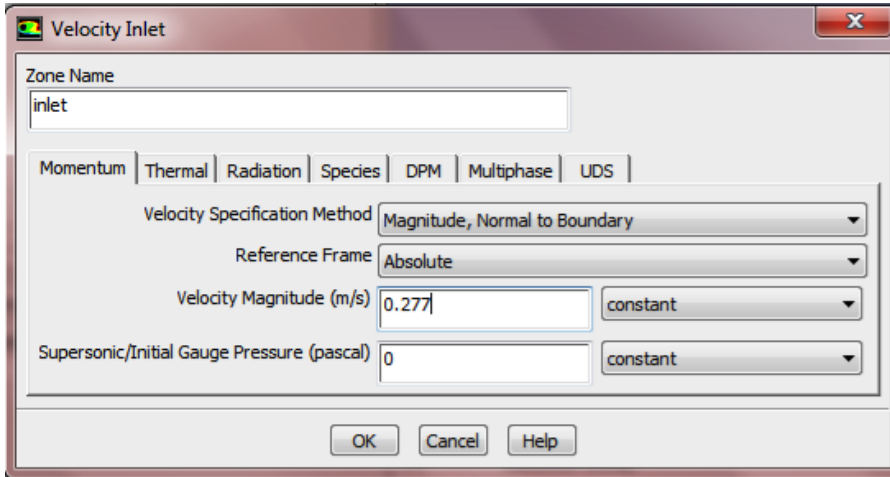
At this point the boundary conditions for the four **Named Selections** will be specified. The boundary condition for the *inlet* will be specified first.

Inlet Boundary Condition

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



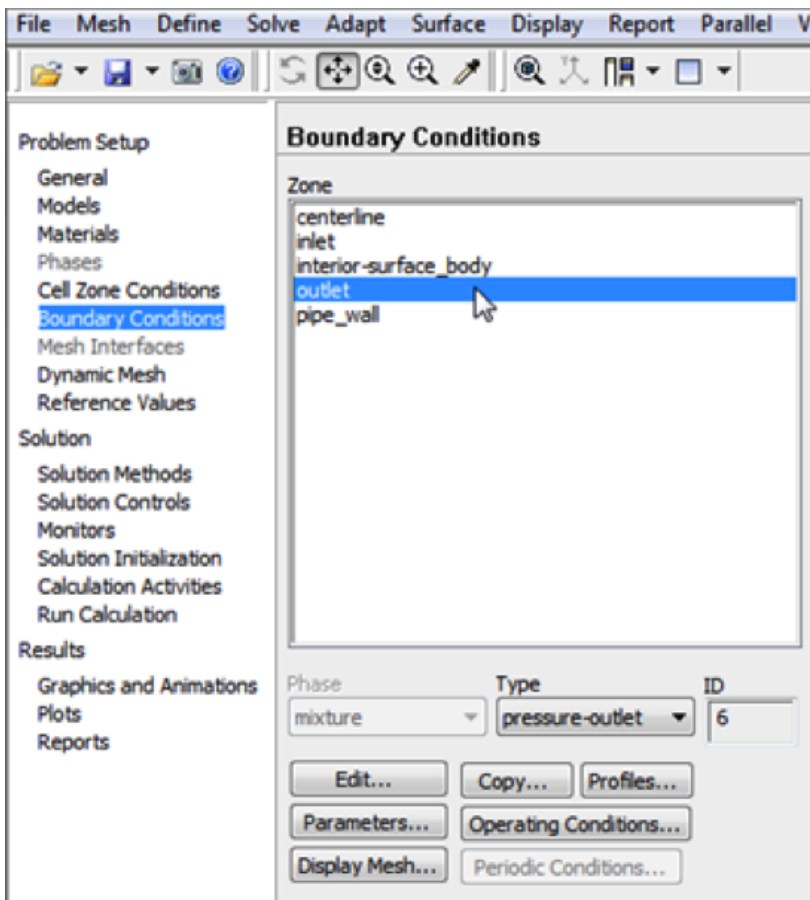
Note that the **Boundary Condition Type** should have been automatically set to **velocity-inlet**. Now, the velocity at the **inlet** will be specified. In the **Velocity Inlet** menu set the **Velocity Specification Method** to **Magnitude, Normal to Boundary**, and set the **Velocity Magnitude (m/s)**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



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

Outlet Boundary Condition

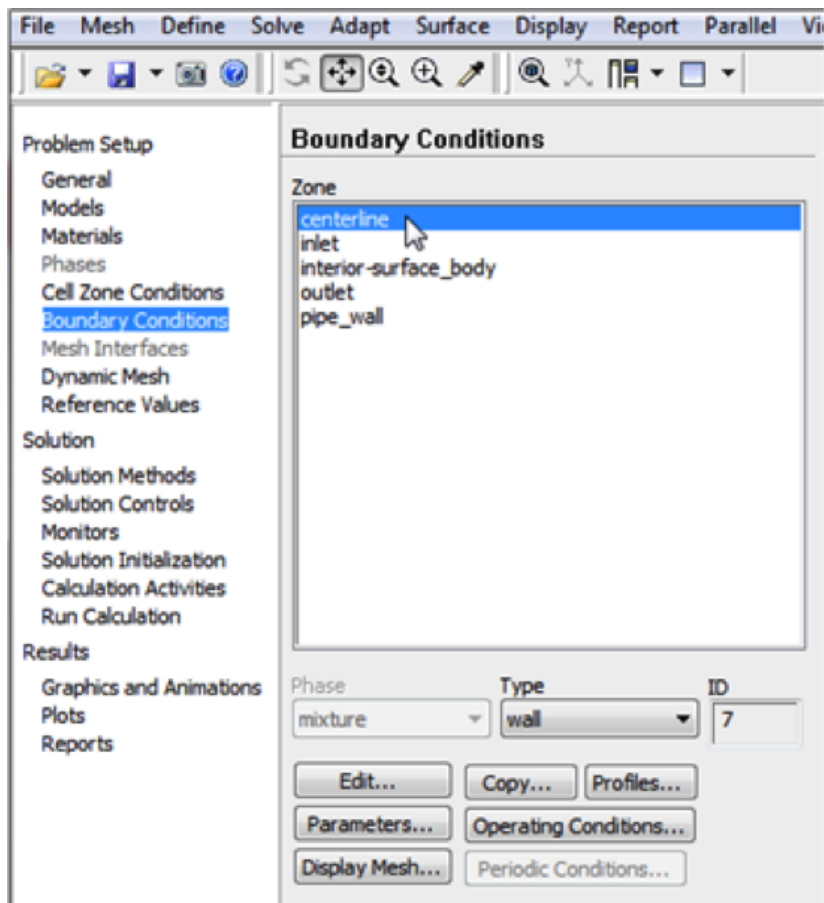
First, select **outlet** in the **Boundary Conditions** menu, as shown below.



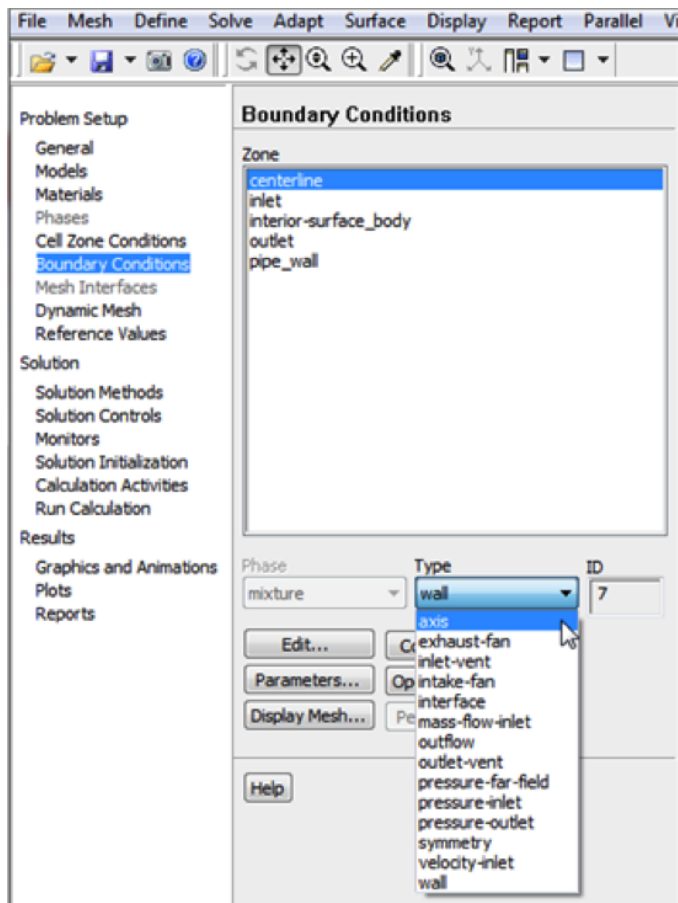
As can be seen in the image above the **Type** should have been automatically set to **pressure-outlet**. If the **Type** is not set to **pressure-outlet**, then set it to **pressure-outlet**. Now, no further changes are needed for the **outlet** boundary condition.

Centerline Boundary Condition

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



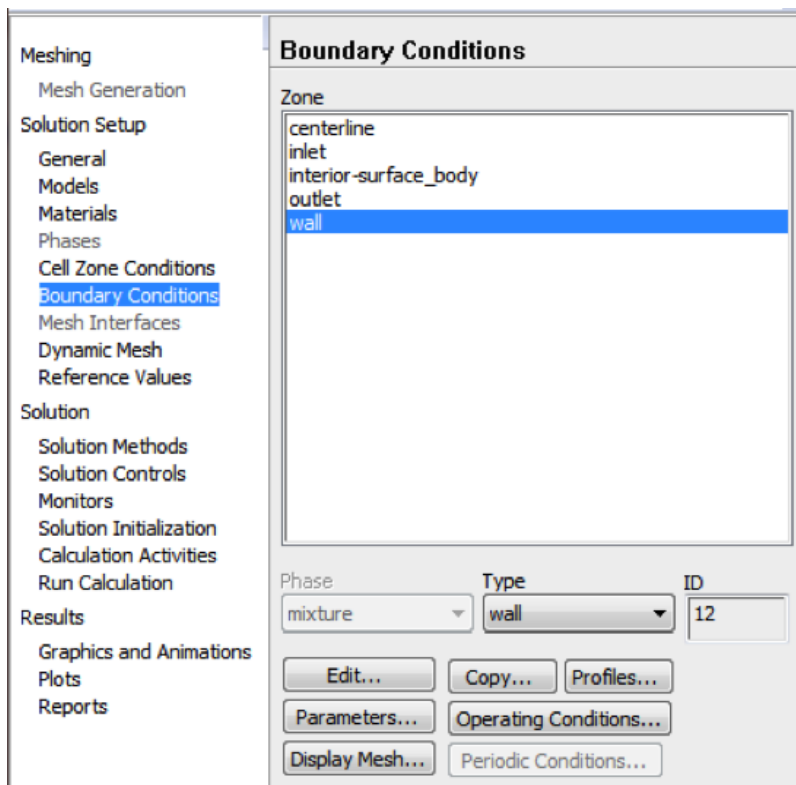
As can be seen in the image above the *Type* has been automatically set to *wall* which is not correct. Change the *Type* to *axis*, as shown below.



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

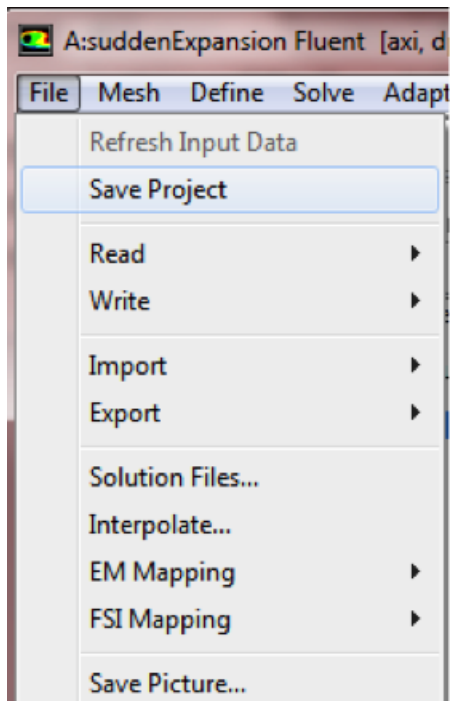
First, select *Wall* in the **Boundary Conditions** menu, as shown below.



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.



[Go to Step 5: Numerical Solution](#)

[Go to all FLUENT Learning Modules](#)