

# Turbulent Pipe Flow (LES) - Physics Setup

Author: Ranjith Tirunagari, Cornell University

[Problem Specification](#)

[1. Pre-Analysis & Start-Up](#)

[2. Geometry](#)

[3. Mesh](#)

[4. Physics Setup](#)

[5. Numerical Solution](#)

[6. Numerical Results](#)

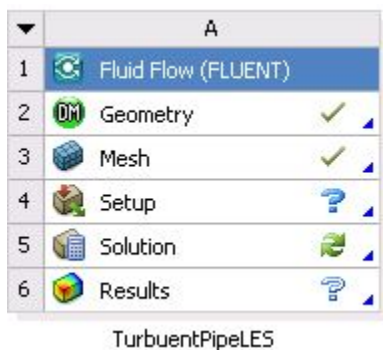
[7. Verification & Validation](#)

[Exercises](#)

[Comments](#)

## Physics Setup

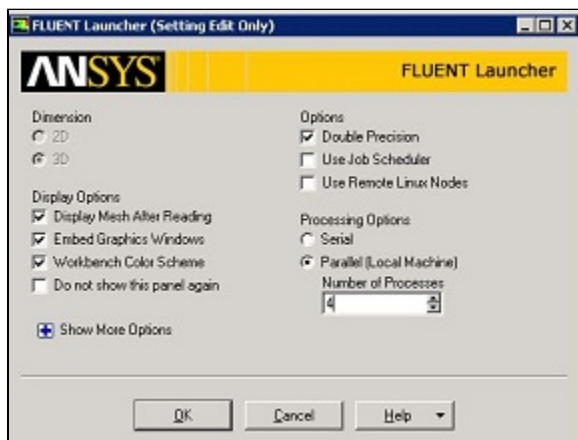
Regardless of whether you downloaded the mesh and geometry files or if you created them yourself, you should have checkmarks to the right of **Geometry** and **Mesh**. Your current **Workbench Project Page** should look comparable to the following image.



A question mark should appear to the right of the **Setup** cell. This indicates that the **Setup** process has not yet been completed. This means that the mesh and the geometry data need to be read into FLUENT.

### 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 Processing Options to "Parallel (Local Machine)" with Number of Processes equal to "4" or to the available number of processors at your end. Click **OK** as shown below.



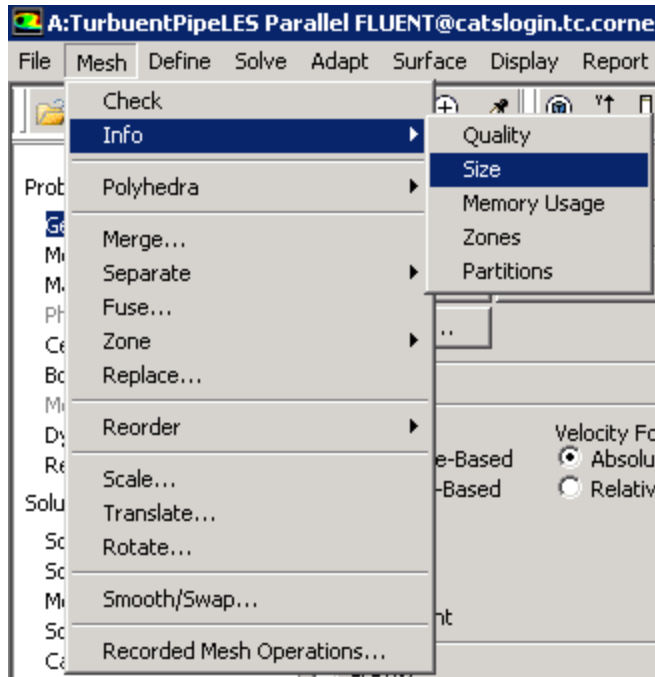
[Higher Resolution Image](#)

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 problem. On the left-hand side of the FLUENT interface, we see various items listed under **Problem Setup** and **Solution**. We will work from top to bottom of both these 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*. (Click) *Mesh > Check* and make sure that the minimum volume is positive. It is a good practice to check if x/y/z - domain extents are according to the dimensions given in the problem specification.

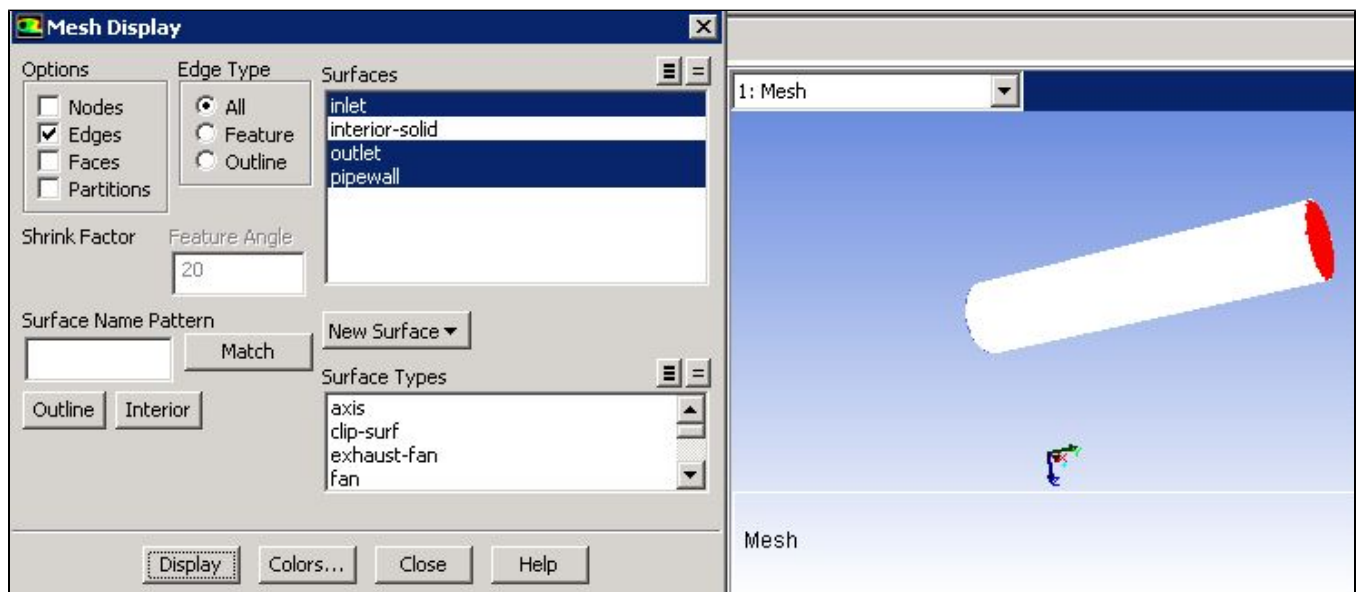
In order to obtain the statistics about the mesh (Click) *Mesh > Info > Size*, as shown in the image below.



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

| Mesh Size                  |        |         |        |            |
|----------------------------|--------|---------|--------|------------|
| Level                      | Cells  | Faces   | Nodes  | Partitions |
| 0                          | 843380 | 1978811 | 332780 | 4          |
| 1 cell zone, 4 face zones. |        |         |        |            |

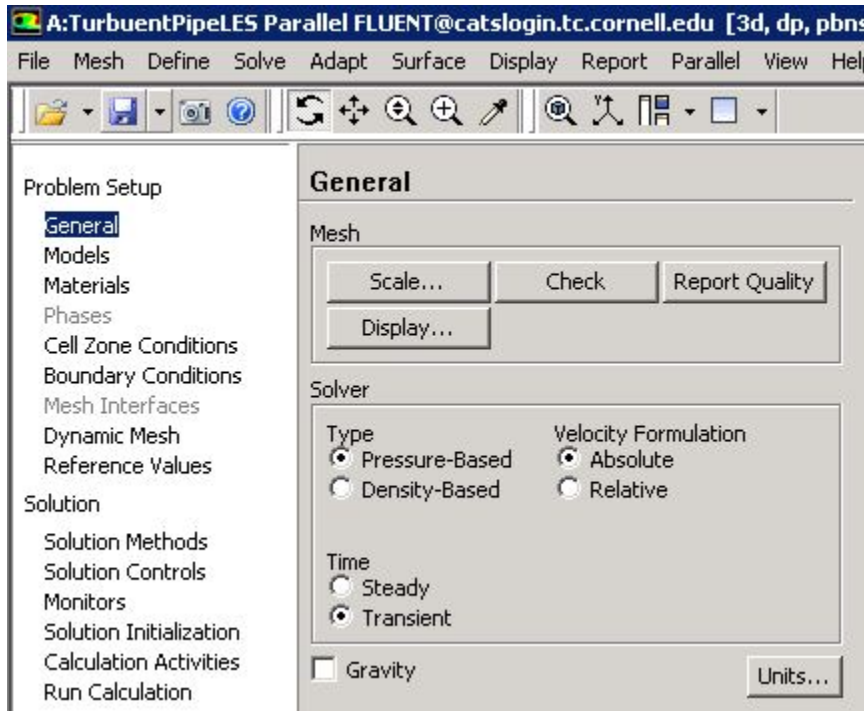
In order to bring up the display options (Click) *General > Mesh > Display*. Make sure you have inlet, outlet, pipewall and interior-solid in *Surfaces* as shown in the figure below.



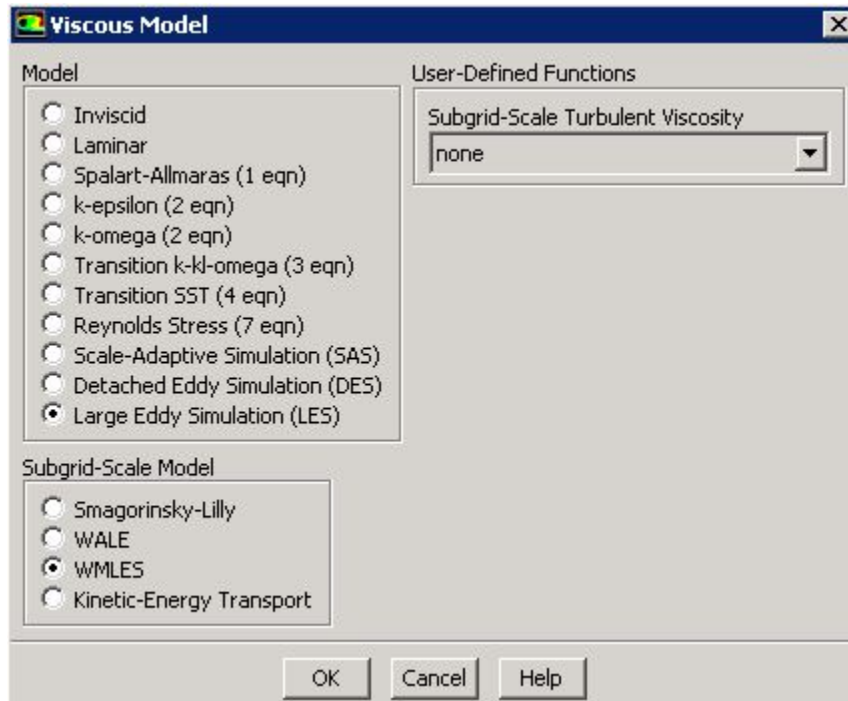
Please review the "Laminar Pipe Flow" tutorial to understand how to rotate, zoom-in and zoom-out the geometry in the *Graphics Window*.

## Define Solver Properties

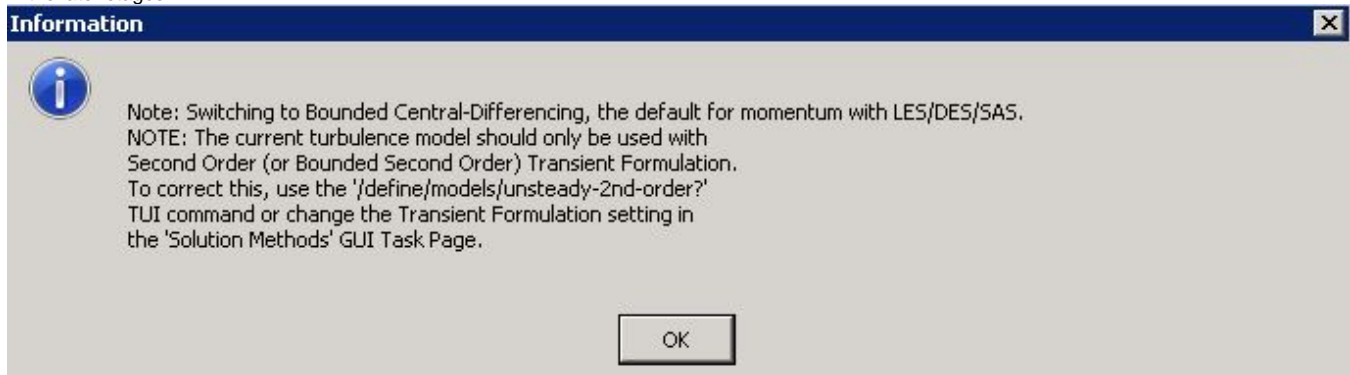
In this section the various solver properties will be specified in order to obtain the proper solution. On the left side of the window (*Click Problem Setup > General*). Make sure that *Pressure-Based* is selected under *Type* and *Transient* is selected under *Time* in the *Solver* section. Note: LES is a transient simulation where the solution is marched in time.



Next, the *Viscous Model* parameters will be specified. In order to open the Viscous Model Options (*Click Problem Setup > Models > Viscous - Laminar > Edit....*). Click *Large Eddy Simulation* under *Model* and *WMLES* under *Subgrid-Scale Model*. Click *OK*.



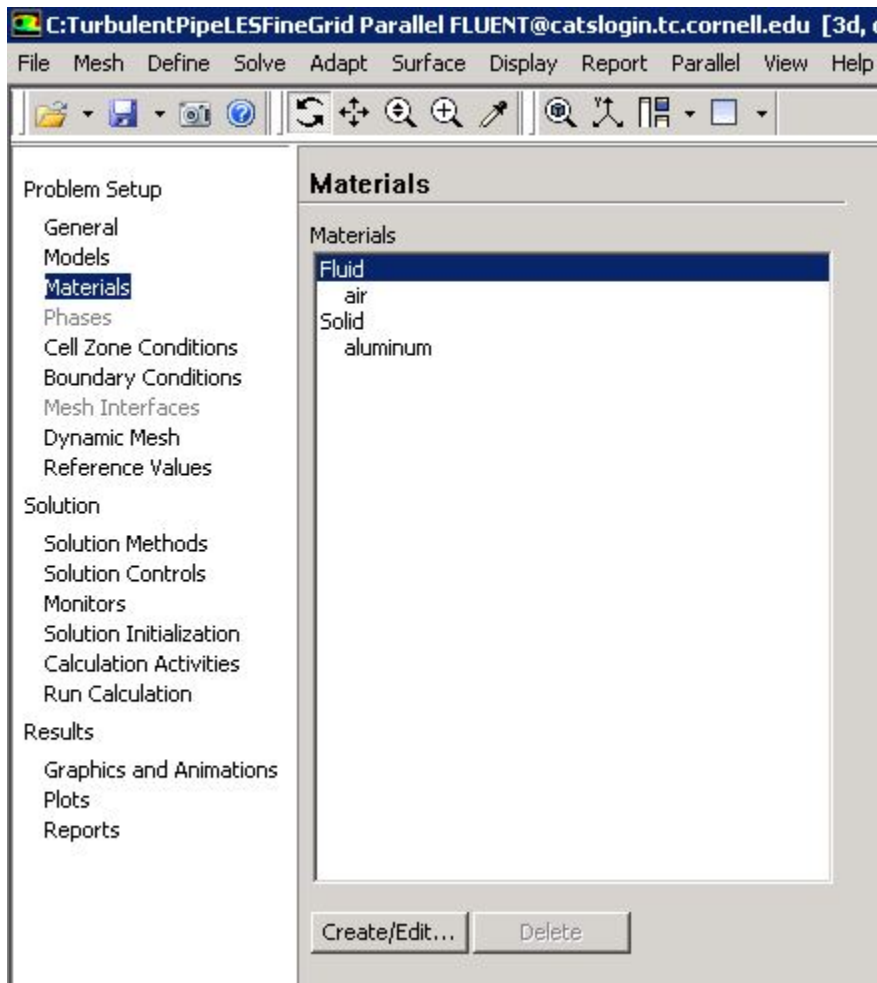
An **Information** box will appear as shown below, click **OK**. Basically, FLUENT switches the discretization scheme for momentum equation to **Bounded Central-Differencing**. It also urges to change the order to **Second Order Implicit** for **Transient Formulation** in the **Solution Methods**, which we will do in the later stages.



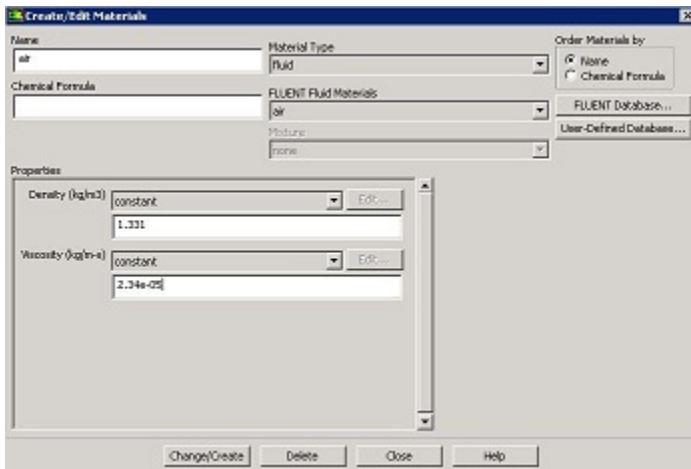
For incompressible flows, the energy equation is decoupled from the continuity and the momentum equations. So the energy equation is not solved. Make sure that **Energy** is set to **Off** in **Problem Setup > Models > Energy**.

### 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**) **Problem Setup > Materials > Fluid > Create/Edit...** as shown in the image below.



In the **Create/Edit Materials** menu set the **Density** to 1.331 kg/m<sup>3</sup> (constant) and set the **Viscosity** to 2.34e-05 kg/(ms) (constant) as shown in the image below.



[Higher Resolution Image](#)

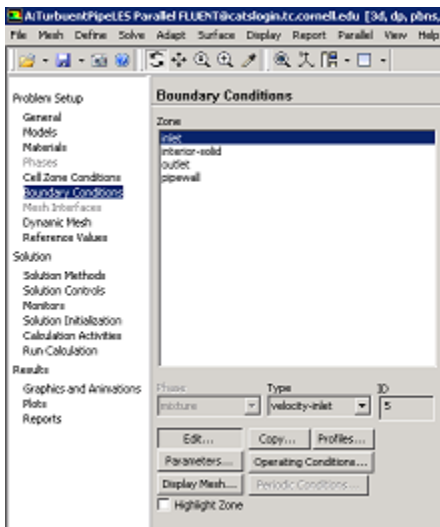
Click **Change/Create**. Close the window.

## Define Boundary Conditions

At this point the boundary conditions for the three **Named Selections** will be specified.

Inlet Boundary Condition

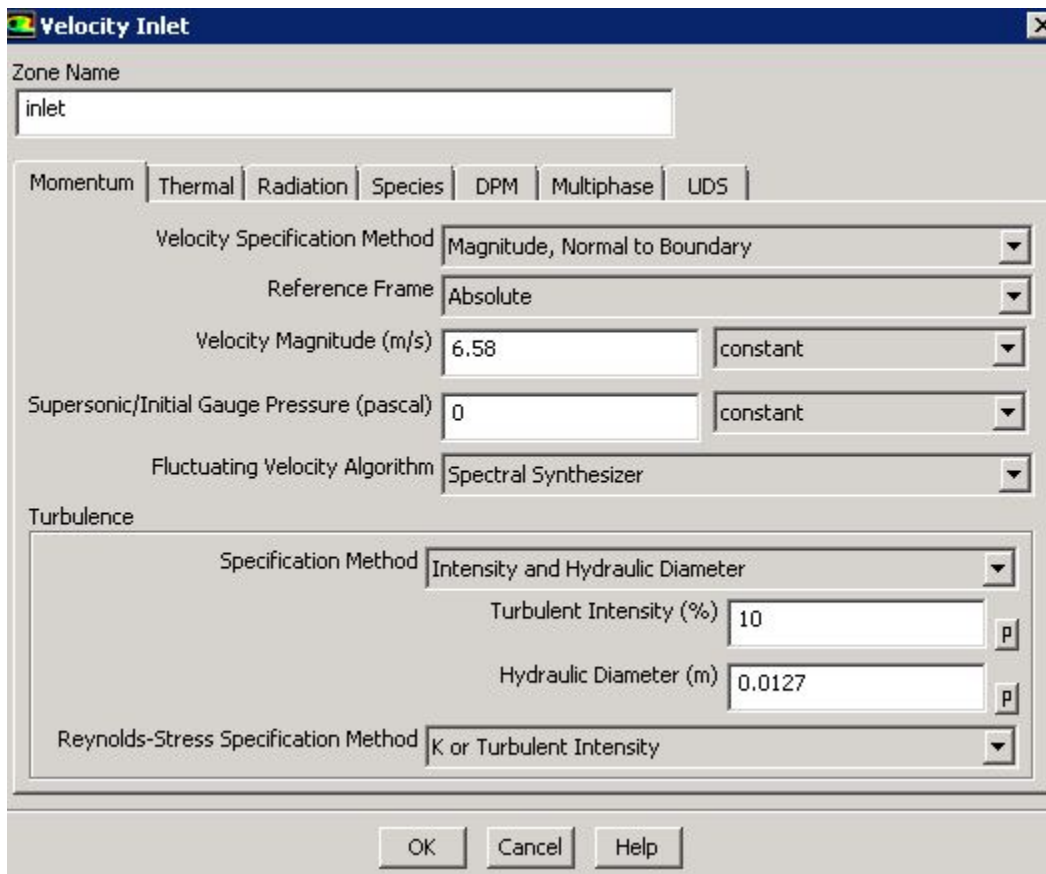
In order to start the process (**Click**) **Problem Setup > Boundary Conditions > inlet > Edit...** as shown in the following image.



[Inlet\\_BC.png](#)

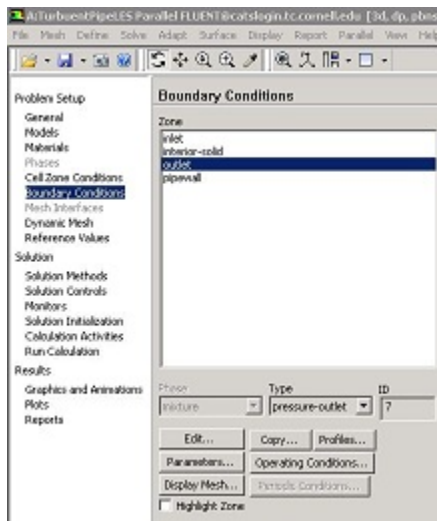
[Higher resolution Image](#)

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**, set the **Velocity Magnitude (m/s)** to 6.58 m/s and set the **Fluctuation Velocity Algorithm** to **Spectral Synthesizer** (this is needed to fluctuate the velocity at the inlet). Also set the **Turbulence Specification Method** to **Intensity and Hydraulic Diameter**. Set the value of **Turbulent Intensity (%)** to 10 % and **Hydraulic Diameter (m)** to 0.0127 m. Finally set the **Reynolds-Stress Specification Method** to **K or Turbulent Intensity** as shown below. Click **OK** to close the **Velocity Inlet** menu.



#### Outlet Boundary Condition

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

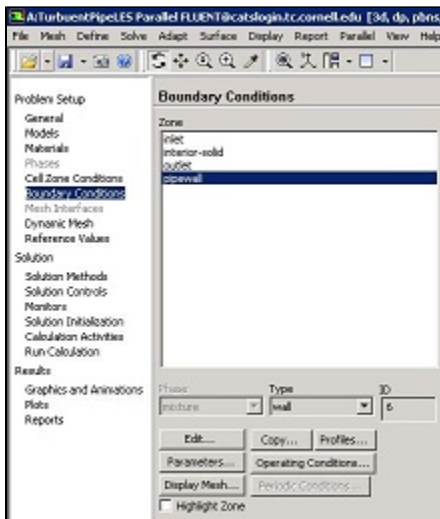


#### Higher Resolution Image

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.

#### Pipe Wall Boundary Condition

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



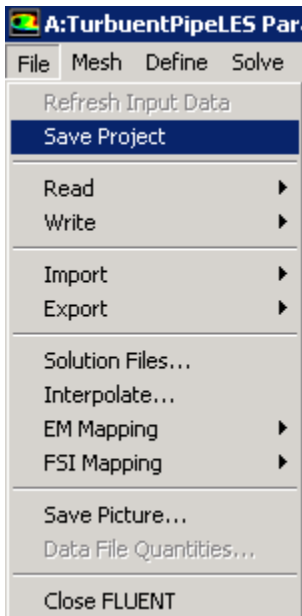
#### Higher Resolution Image

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 **pipe\_wall** boundary condition. Also make sure that the boundary condition **Type** for **interior-solid** is **Interior**.

It is a good practice to change the **Reference Values** now, these values can be useful when we are postprocessing results later on. **(Click) Problem Setup > Reference Values**. Select **Compute from** as **inlet**.

#### 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](#)