

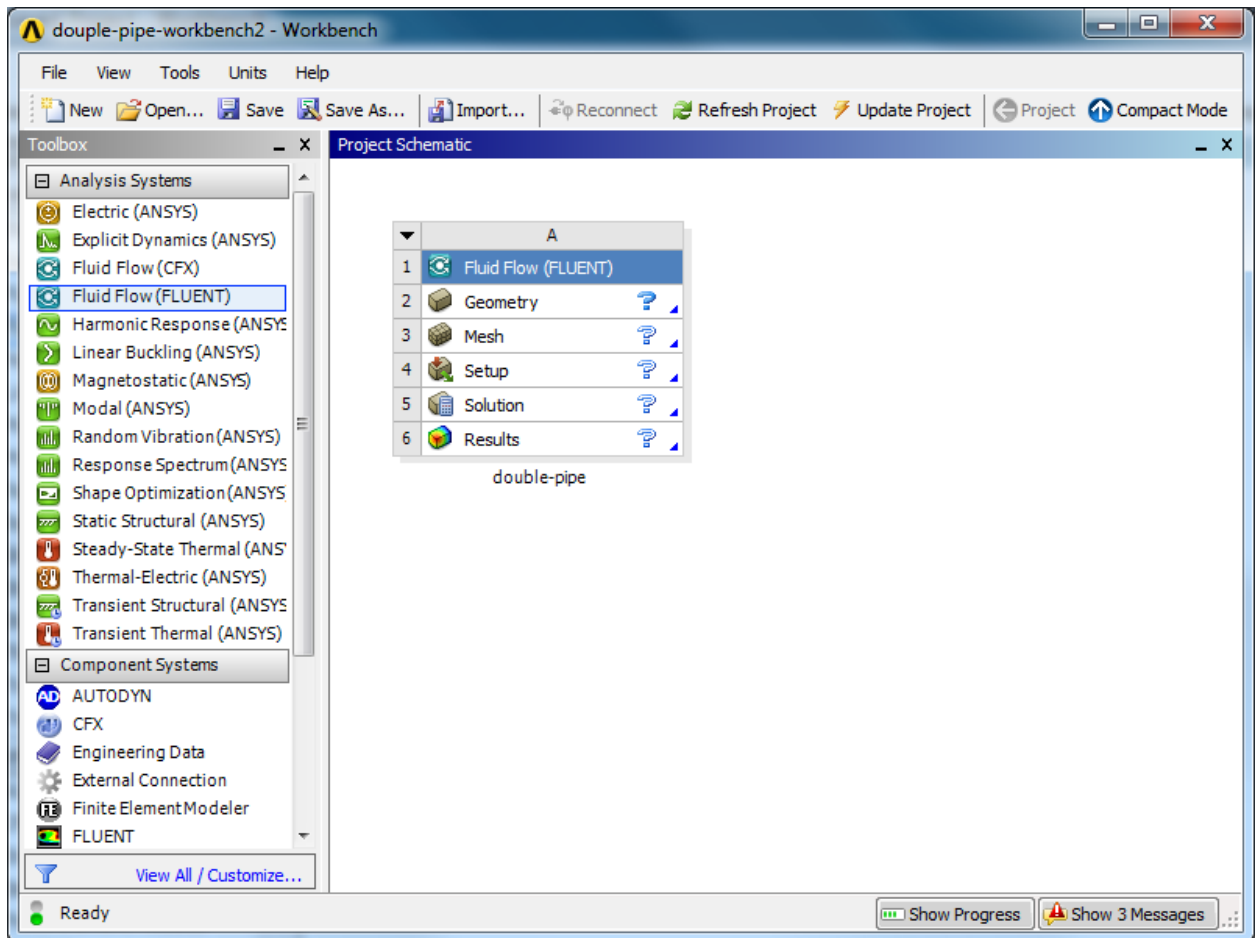
Tutorial for laboratory project #2

Using ANSYS Workbench

For Double Pipe Heat Exchanger

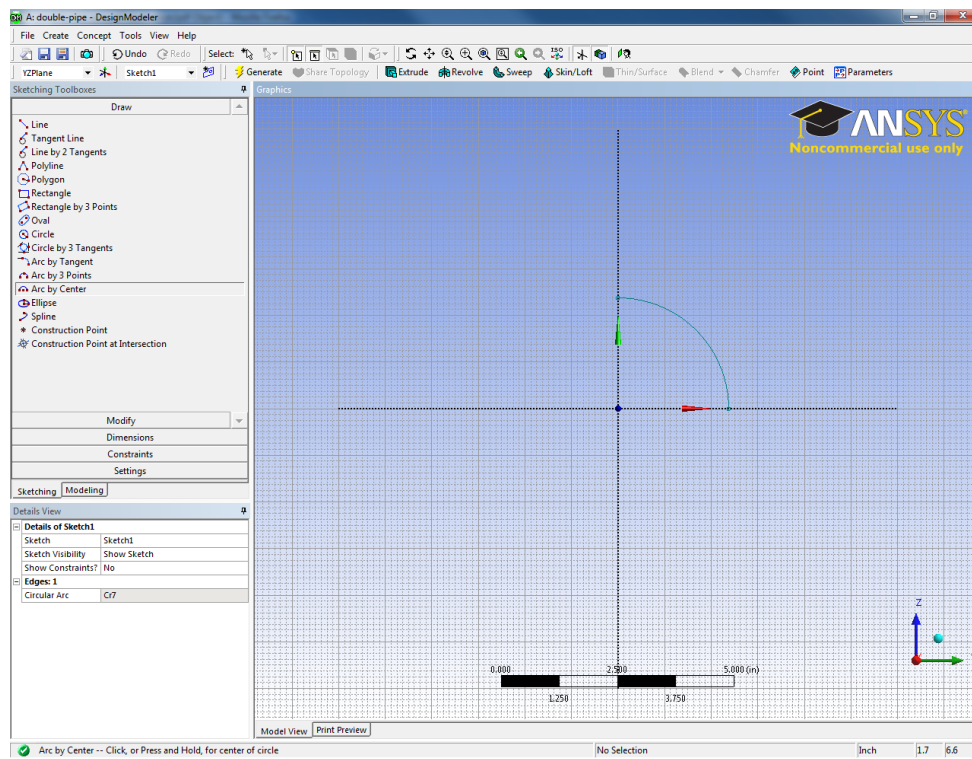
1. Preparing ANSYS Workbench

- Go to Start Menu/All Programs/Simulation/ANSYS 12.1/Workbench.
- In the toolbox menu in the left portion of the window, double click **Fluid Flow (Fluent)**. A project will now appear in the project schematic window of Workbench.
- Right-click **Fluid Flow (Fluent)** under the **Project Schematic** and select **Rename**. And enter double-pipe for the name of your project.
- Click **Save** button in menu bar to save the project. A **Save As** window will pop up. Enter double-pipe-workbench in **File name** as a name of your workbench and click on **Save** button. A new file of double-pipe-workbench.wbpj will be added to the **Files**.
- The default working environment for Geometry is 3-D. So you don't need any action for this.

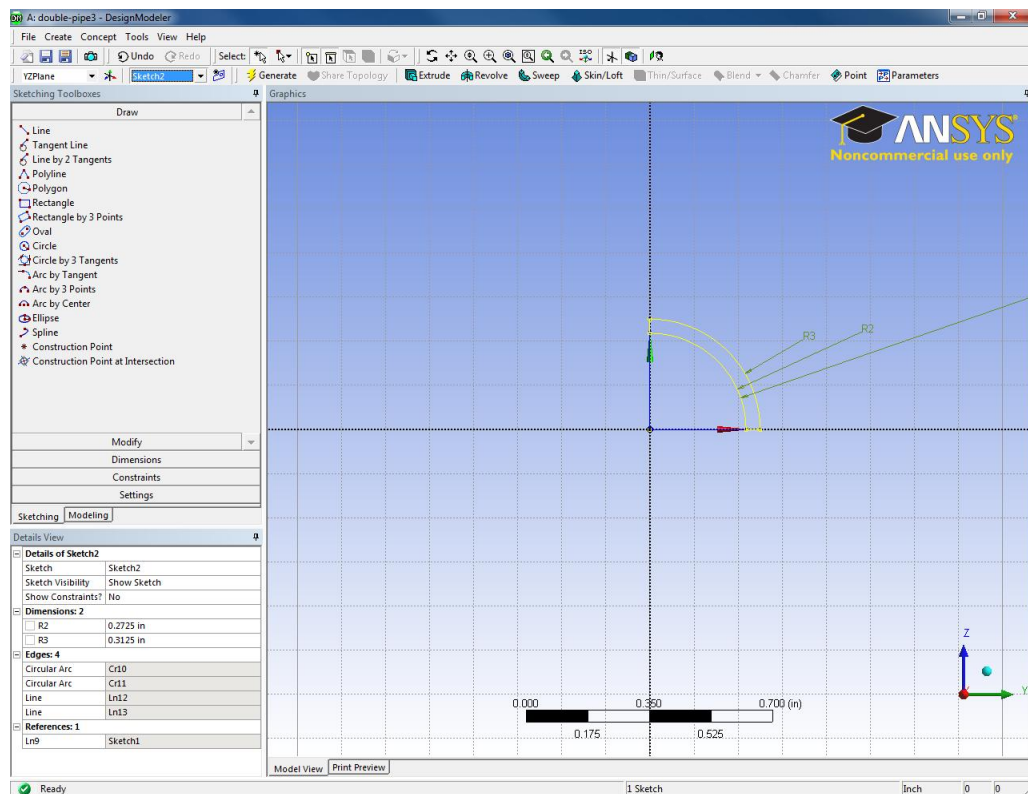


2. Creating Geometry

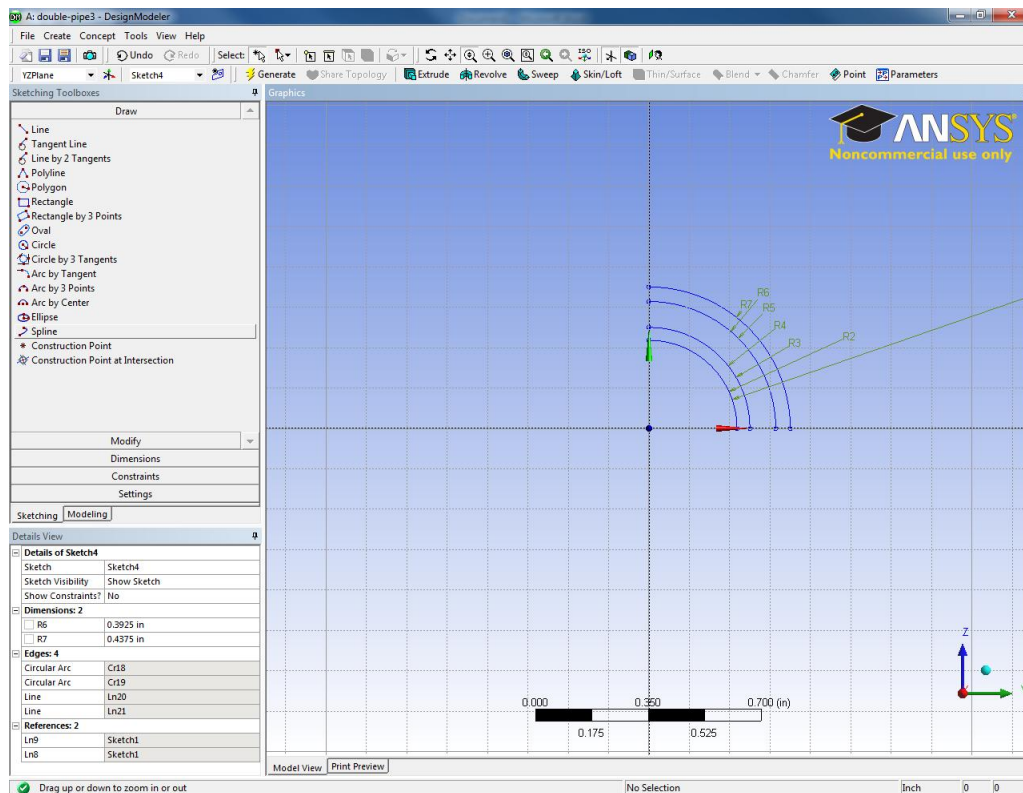
- In the **Project Schematic**, double-click the **Geometry**. This will open **ANSYS DesignModeler** (This will be a logo with a green DM).
- Select **Inches** as the unit type, and click **OK**.
- In the **Tree Outline**, right-click **YZPlane** and select **Look At**. This will orient the view to be normal to the XY plane for 2-D.
- Click **New Sketch** button from the **Plane/Sketch** toolbar. (You can find the **New Sketch** by moving the mouse cursor). Sketch1 under the YZPlane will be created.
- Repeat this three times to create **Sketch2, Sketch3, and Sketch4**
- Select **Sketch1**, and click the sketching tab at bottom of the tree outline window.
- Click **Settings, Grid**, and select **Show in 2D** and **Snap**. Define the **Major Grid Spacing** to be 1 inch, and set **Minor-Steps per Major** to be 8. This defines the large grid to be 1 inch, and the small grid to be 1/8 inch.
- Click **Draw**, and select **Arc by Center**. You can scroll down the scroll bar. Click the origin (0,0) to define the center of the circle, then click the Y axis to define the start of the arc, then the Z axis to define the end.



- Select **Line** from the **Draw** menu by scrolling up under sketching toolboxes. Click the one end of the arc, and then click the origin to form a line. Repeat this to form a closed quarter circle.
- Click **Dimensions**, then **Radius**. Select the arc in Graphics window, then click again to place the dimension.
- In Details View, change the value of R1 to be .2725 (inches).
- In the Sketching Toolboxes window, click the **Modeling** tab.
- Select **Sketch2**, and return to the **Sketching** tab.
- Using the **Arc by Center**, create two arcs in a similar way as in Sketch1. The second arc should be larger to avoid confusion.
- Select **Line**, and connect both the respective arc ends.
- Selection **Dimensions**, then **Radius**. Create radial dimensions for the two arcs in Sketch2, starting with the inner most arc.
- In Details View, set R2=.2725, and R3=.3125

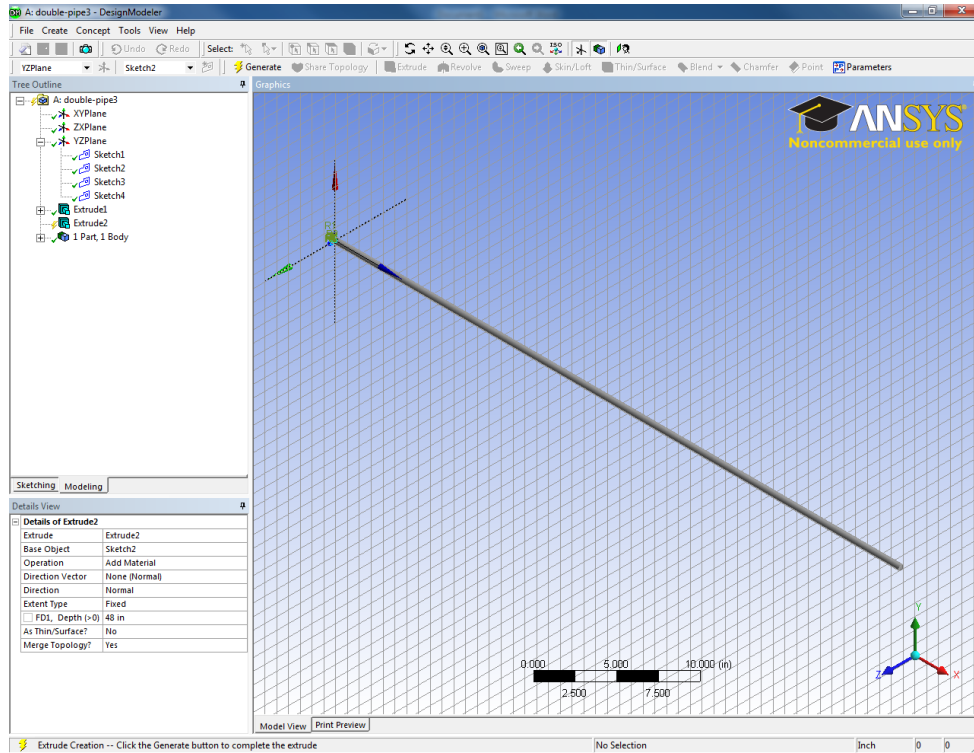


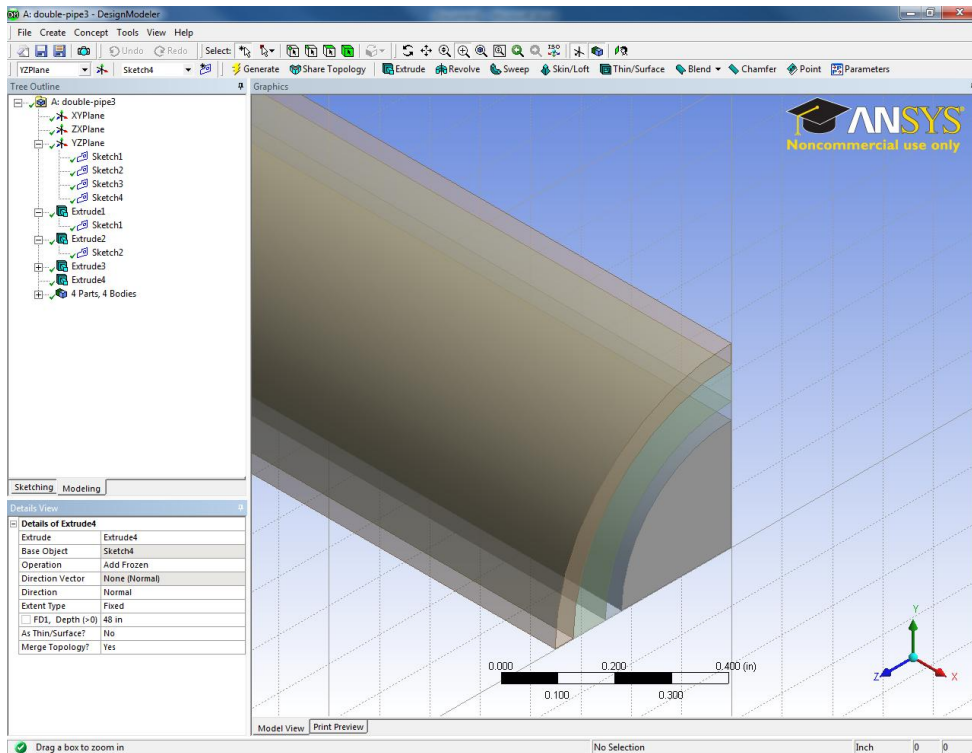
- Select **Sketch3** from the **Modeling** tab and return to **Sketching**.
- Create two arcs using the same method, join them using **Line**, and set $R4=.3125$, and $R5=.3925$ in **Details View**.
- Select **Sketch4** from the **Modeling** tab and return to **Sketching**.
- Create two arcs using the same method, join them using **Line**, and set $R6=.3925$, and $R7=.4375$ in **Details View**.
- Return to the **Modeling** tab.
- Your screen should be similar to the following:



- Select **Sketch1**, then select **Extrude** from the **3D Features** toolbar. It will create **Extrude1** with a light bolt symbol under the **Tree Outline** window
- Change the **Depth** to 48 (inches) under **Details View**.
- Click **Generate**, then **Iso**. You should see the innermost section extruded in the positive X direction.
- Select **Sketch2**, then select **Extrude** from the **3D Features** toolbar.
- Change the **Operation** to **Add Frozen** in **Details View**. Make sure the **Depth** is 48. The Add Frozen allows to create a separate body without merging.

- For **Sketches 3 and 4**, repeat the **Extrude** command making sure to use the **Add Frozen** operation.
- Click the ball in the center of the coordinate axes to see the isometric view. Using **Box Zoom** in the tool bar, enlarge the end part of the pipe by clicking and dragging, as shown in the following figure.

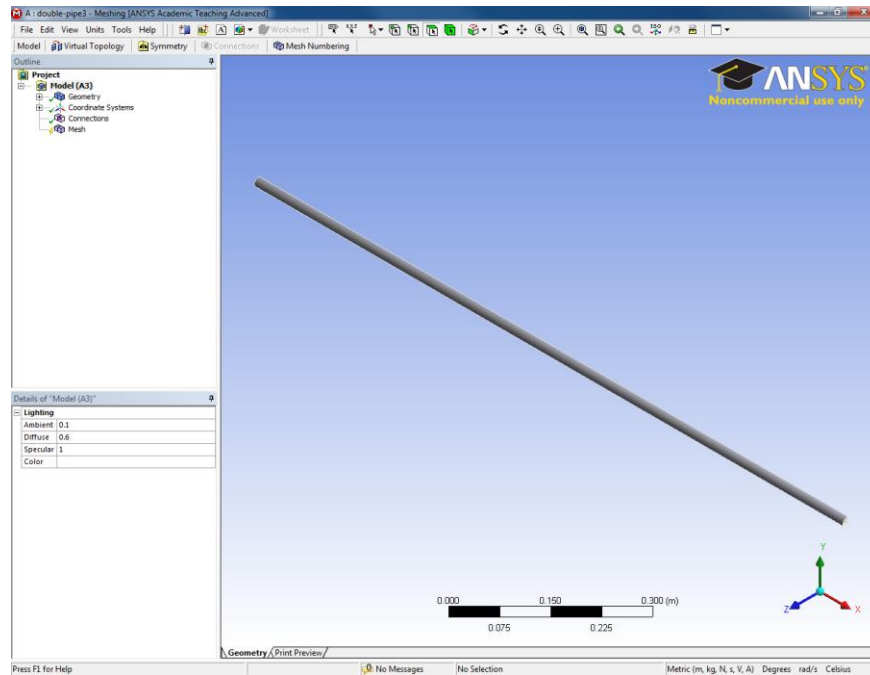




- Verify that in the **Tree Outline** that it says 4 Parts 4 Bodies, and expand the listing. Click the first body among the four bodies.
- In **Details View**, change body (name) to inner_fluid and change **Fluid/Solid** to **Fluid**.
- Select the second body and change the name to inner_pipe, and leave **Fluid/Solid** as **Solid**.
- Select the third body, change name to outer_fluid, and change **Fluid/Solid** to **Fluid**.
- Select fourth body, change name to outer_pipe, and leave **Fluid/Solid** as **Solid**.
- Hold down the Ctrl key, select all four named bodies, and right-click on them. Select **Form New Part**.
- Go to **File**, Close **DesignModeler**.
- Save in Workbench.

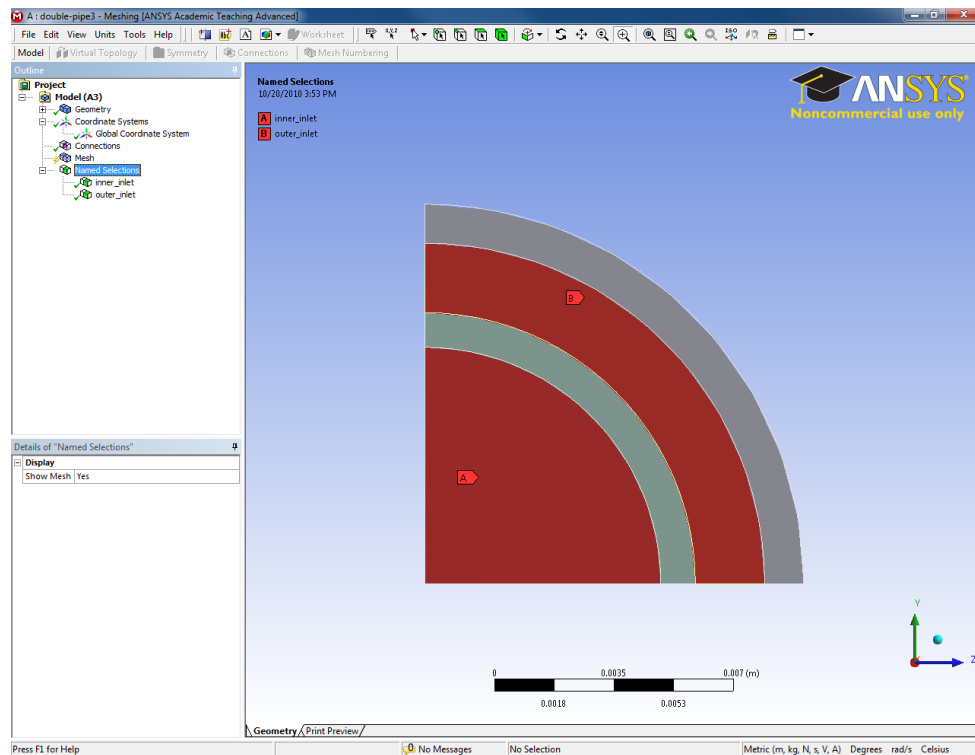
3. Mesh Generation

- In the **Project Schematic**, double-click **Mesh**. It may take a minute or two to load.
- Click **OK** in the **Meshing Options** window.



(a) Name the surfaces for the boundary conditions.

- Move the cursor between the Y and Z axis on the axis triad. A black arrow labeled **-X** will appear, click on it. It will show a YZPlane view.
- Right-click an arbitrary portion of the screen, and change the Cursor Mode to **Face**.
- Click the innermost section (inner fluid), then right-click on it and select **Create Named Section** and enter inner_inlet.

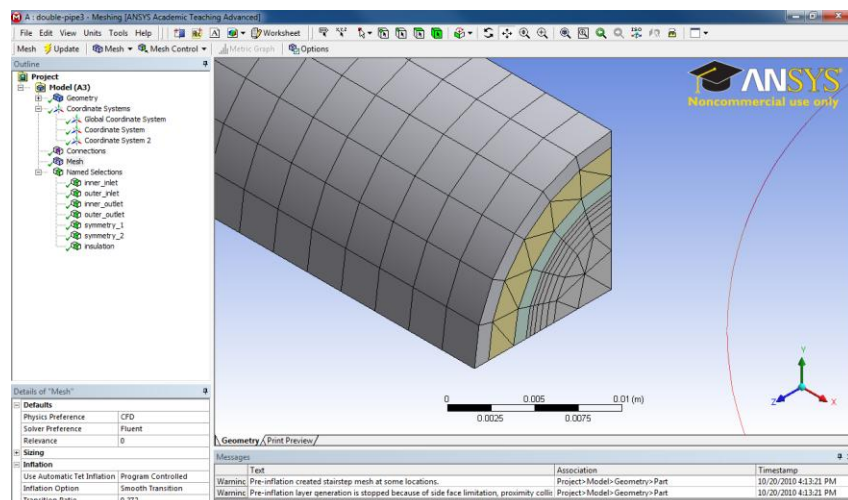


- Click the third section (outer_fluid), then right-click on it, and select **Create Named Section** and enter outer_inlet.
- Click the center ball on the triad to have the isometric view.
- Click the +X axis on the triad to see the other side of the double pipe.
- Click the innermost section, then select **Create Named Section** and enter inner_outlet.
- Expand the **Named Selections** in the **Outline**, this will create labels for the named faces.
- Click the center ball on the triad again.
- Click the outermost surface of the pipe.
- Right-click and select **Create Named Section** and enter insulation_surface.
- Move the cursor between the X and Y axis on the axis triad. A black arrow labeled -Z will appear, click on it. It will show a XYPlane view.

- Click Box Zoom in the Toolbar and zoom in any section of pipe.
- Holding down the Ctrl key, select all four faces. Right-click and select **Create Named Section** and enter symmetry_1.
- Move the cursor between the X and Z axis on the axis triad. A black arrow labeled -Y will appear, click on it. It will show a XZPlane view.
- Click Box Zoom in the Toolbar and zoom in any section of the pipe.
- Holding down the Ctrl key, select all four faces. Right-click and select **Create Named Section** and enter symmetry_2.

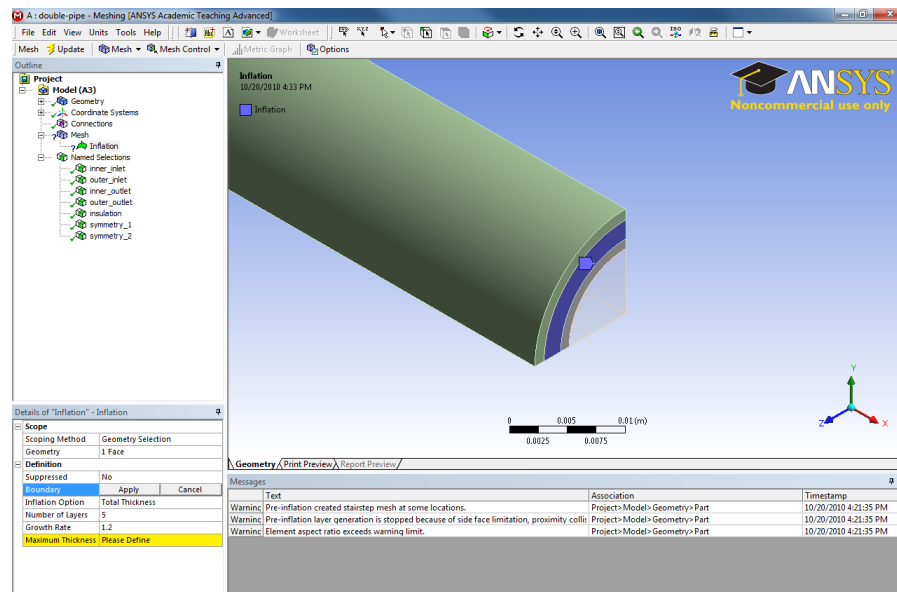
(b) Create meshing

- Select **Mesh** in the **Outline** window by clicking it.
- In the **Details of “Mesh”** window, expand the **Sizing** option and change the **Relevance Center** from **Coarse** to **Medium**.
- Expand the **Inflation** option.
- Set **Use Automatic Tet Inflation** from **None** to **Program Controlled**.
- In the **Outline**, right-click **Mesh** and select **Update**. This step will create meshing and take several minutes. The meshing is shown in the following after zooming in.



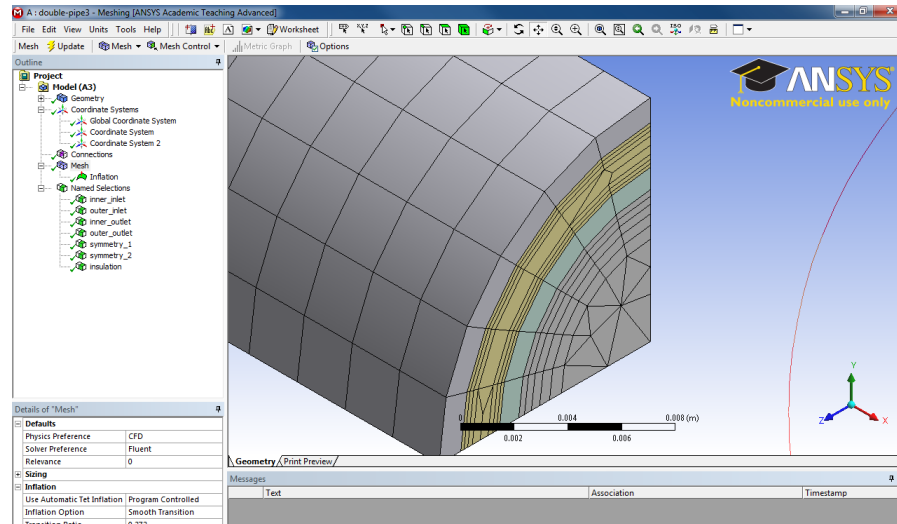
(c) Correction on Inflation

- The meshing in the inner fluid appears adequate, while the meshing in the outer fluid does not.
- Change the **Cursor Mode** to **Face** by right-clicking
- Click the face of the outer_fluid.
- Click **Mesh Control** in the Toolbar, and select **Inflation**.



- Change the **Cursor Mode** to **Edge**.
- Hold down Ctrl key, and select the two curvature edges of the outer_fluid. Click **Apply** in **Boundary** under **Definition** of the Detailed "Inflation" window.
- Change the **Number of Layers** to 3.
- Change the **Maximum Thickness** to 0.001.
- Click **Update**.
- **File/Close Meshing**.
- **SAVE**

- The final meshing is shown in the following.

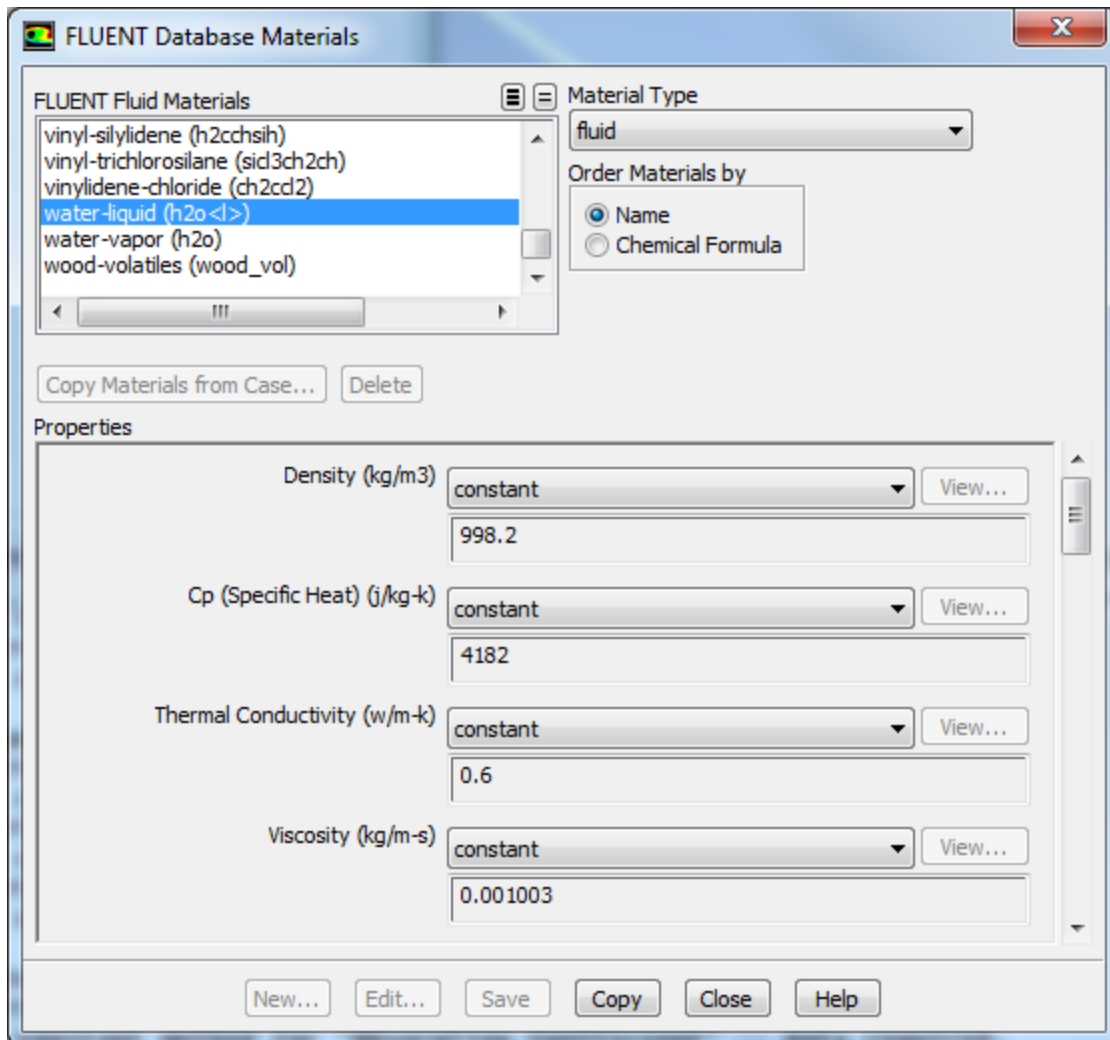


4. Fluent Setup

- In Workbench, double click **Setup**.
- Click **OK** to FLUENT launcher prompt.
- In General, click **Units** and select **Length** from the list by scrolling, and change to **in** (inches), then **Close**.
- Select **Models**. Double-click **Energy** to **On**. Double-click **Viscous**, set type to **k-Epsilon (2 eqn)**, and hit **OK**.
- Double click **Radiation**, and change to **Discrete Ordinates (DO)**. Hit **OK**.

(a) Setup Materials

- Select **Materials**.
- Click **Create/Edit**, then click **Fluent Database**.
- Select **water-liquid (h2o)** from the list. Click **Copy**, then **Close**. Close again.



- Click **Create/Edit**, then **Fluent Database**.
- Change Material Type to **Solid**.
- Select **Copper**, hit **Copy**, then **Close**. Close again to return to main window.

(b) Setup Cell Zone Conditions

- Select **part-inner_fluid**, and click **Edit**. Change **Material Name** to **water-liquid**. And click **OK**.
- Select **part-inner_pipe**, and click **Edit**. Change **Material Name** to **copper**. **OK**.

- Select **part-outer_fluid**, click **Edit**. Change **Material Name** to **water-liquid**. **OK**
- Select **part-outer_pipe**, click **Edit**. Change **Material Name** to **copper**. **OK**

(c) Select Boundary Conditions.

- Select **inner_inlet** from **Zones**, and click **Edit**.
- In the **Momentum** tab, change **Velocity Magnitude** (m/s) to 0.8384, which is the value based on 2 gallon per minute.
- Change the **Turbulent Kinetic Energy** to 0.01 and change the **Turbulent Dissipation Rate** to 0.1.
- Select the **Thermal** tab, and change **Temperature** to 288 K, which is the value entering inner pipe as a cold water. Click **OK**.
- Repeat this for **outer_inlet** from **Zones**, and click **Edit**.
- In the **Momentum** tab, change **Velocity Magnitude** (m/s) to 0.9942, which is the value based on 1.8 gallon per minute.
- Change the **Turbulent Kinetic Energy** to 0.01 and change the **Turbulent Dissipation Rate** to 0.1.
- Select the **Thermal** tab, and change **Temperature** to 323 K, which is the value entering outer_pipe as cold water. Click **OK**.
- In the **Zone** list, verify that the symmetry zones are the symmetry type. And also make sure that the insulation_surface has zero heat flux as default.

(d) Obtain the Solution.

- Select **Solution Methods**.
- Change **Momentum**, **Turbulent Kinetic Energy**, and **Turbulent Dissipation rate** to **Second Order Upwind**.

- Select **Solution Initialization**, and click **Initialize**.
- Select **Run Calculation**, and set the **Number of Iterations** to 650.
- Click **Calculate**. This step will take a while to compute the results.

(e) Viewing Results

- When the computation converges, click **Graphics and Animations** under **Results**.
- Double-click **Contours**.
- Enable **Contours of Temperature, Static Temperature**. Click **Filled** under **Options**. Set the **Number of Levels** to 40. Select all of the **Surfaces**, except those which are **interior**. And click **Display**. The view will be small and long.
- On the Menu bar, select **Display/Views/Camera**.
- Change **Projection** to **Orthographic**. You can now rotate the part around for better viewing as shown in the following.

