



### Mixing Tee Revisited

In the ANSYS Workbench, create a new “Fluid Flow (Fluent)” block. Open “Geometry” to create the mixing tee. *You should retain this workbench project for a future lab exercise.*

- In the first dialog box, select “Centimeter” as the base length unit.
- In the left-hand *Sketching Toolboxes* panel, select “XYPlane” under the top-level node in the tree.
- Next, select the *Sketching* tab.
  - In the *Settings* panel, check the *Grid* option “Snap” and set the number of minor intervals to “2”. (This allows us to use half-centimeter increments in our plotting.)
  - In the *Draw* panel, select “Rectangle”.
  - Position your mouse cursor at (0, -15), click, and drag to (2.5, 15). This creates a rectangle sketch, which we will rotate around the  $y$ -axis to create a pipe. (Make sure the coordinates are exact—it is difficult to move a sketch object in DesignModeler, and often easier to delete a sketch and draw it again.)
  - We will now create a new sketch by selecting the *New Sketch* button on the toolbar.
  - With the new “Sketch2”, repeat the forgoing process for a rectangle from (0, 0) to (-15, 1.5), which we intend to rotate around the  $x$ -axis to represent the hot-water inlet pipe.
- Select the *Modeling* tab. “Sketch1” and “Sketch2” should both appear under “XYPlane” in the tree. We will now generate these as objects.
  - Click on “Sketch1” and press the *Revolve* button on the toolbar. 
  - Click “Appl” for “Geometry”.
  - Select “Axis”. Select the  $y$ -axis and then click “Apply”.
  - A small lightning bolt ⚡ has appeared next to the new entry “Revolve1” in the tree. This universally means that you need to press the *Generate* button . Do this now.
  - A cylinder appears which intersects the sketched rectangle, indicating that we have successfully created this object.
  - Repeat this process for the hot-water inlet pipe, making sure to rotate it around the  $x$ -axis instead of the  $y$ -axis.
- We now have a complete geometry. Save this project and return to the *ANSYS Workbench*.

