

Solidworks Practice Homework

Place in public directory by beginning of Class III

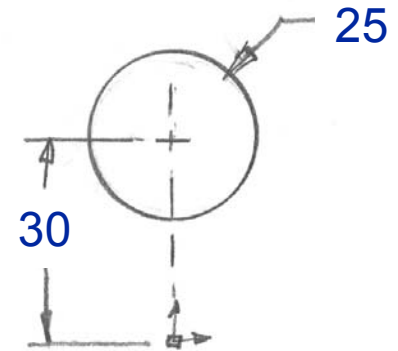


Solidworks Practice

Housing



- Open a new part file
- Create Cylinder
- Open a sketch on the **Front (or plane 1)** Plane
- Draw a Construction line up from the origin
- Draw a circle with the center on the construction line, and dimension it to be **25mm**
- Add another Dimension from the origin to the circle center. Make it **30mm**
- Select **Extrude Feature**, and select **Mid Plane** and enter **80mm**
- Label this *Cylinder*



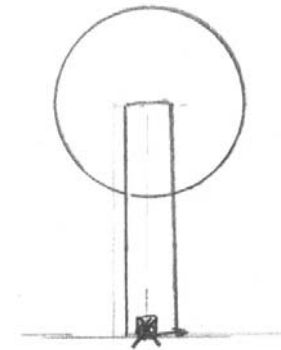
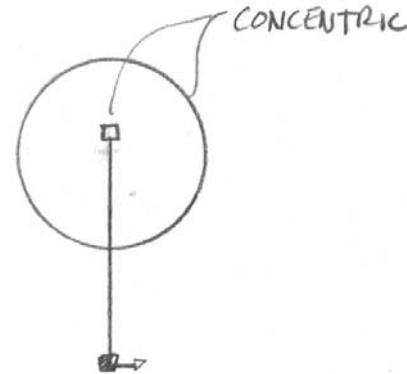
Solidworks Practice

Housing



Create Vertical Bar

- Open a sketch on the face of the cylinder
- Draw a vertical line up from the origin and through the **center** line of the *Cylinder*
- Click off the line, click on the top endpoint, and **Add a relation** to make it **concentric** with *Cylinder*
- Hit **Extrude Boss/Base** button
- On the **Thin Feature** Tab, Select Mid-Plane Type, and **7.5mm** Wall Thickness
- Use the blind condition with 80mm setting.
- Click **OK** and label this *Vertical Bar*



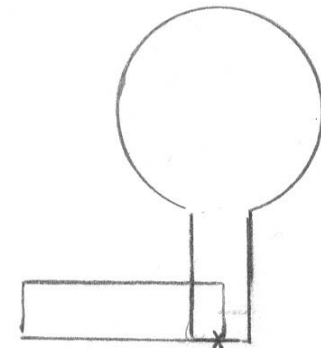
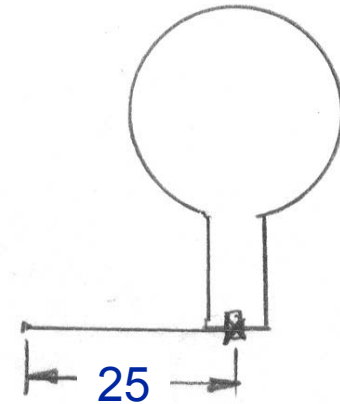
Solidworks Practice

Housing



Create the Base

- On the face of the cylinder, **open a sketch** and draw a **line** from the **origin** to the left
- Dimension it **25mm**.
- Hit the **Extrude Base** Button; Hit the **Thin Feature** Tab and select a • Wall Thickness of 7.5mm and **Type: One-Direction**. Make sure it is thickening upward. Again use Blind, 80mm.
- Label this *Base*



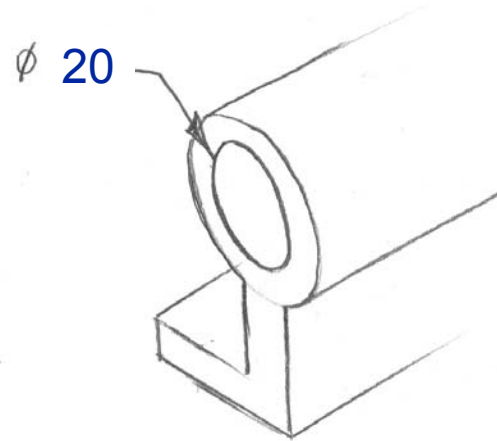
Solidworks Practice

Housing



Make inside hole

- Select the front face of the part and open a sketch
- Orient the view towards **Front** (from the standard views toolbar)
- Draw any Circle and Dimension it to **20mm** diameter
- Hold down CTRL, Select the outer edge of the *Cylinder* feature, and right click to **Add Relations**, and choose **Concentric**
- ^a Select **Extrude Cut Feature**, and select **Through All** in the **Type** box
- Click OK and label this *Hole*

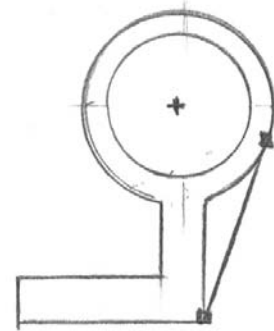


Solidworks Practice

Housing



Create a Rib



- Select the **Front Plane (or plane 1)** and open a sketch
- On the Front view, draw a line from the bottom corner of the 'L' to a point on the outside *Cylinder* edge. Make sure that you are on the center plane.
- Highlight both the line and the *Cylinder* edge, and right click **Add Relations**, selecting **Tangent**
- Select **Insert, Features, Rib**
- The **Property Manager** appears. Select **Mid plane Thickness** of 7.5mm. For **Extrusion Direction**: Parallel to sketch. Toggle the **Flip Material Side** box.
- Select **OK** and label this *Rib*

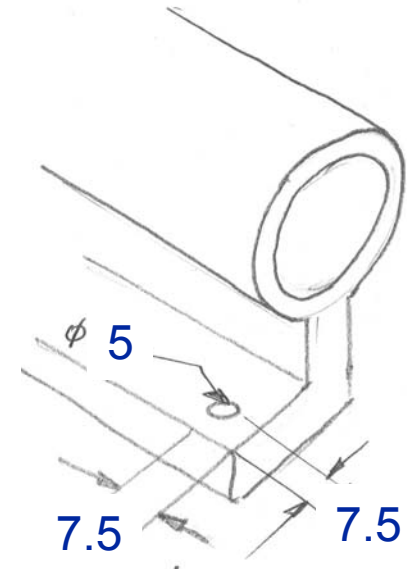
Solidworks Practice

Housing



Create a Mounting Hole

- Highlight the top plane of the *Base* and open a sketch
- Draw a circle, and Dimension it 5mm diameter
- Dimension it 7.5mm away from the Front face, and 7.5mm away from the side face furthest from the origin
- Select **Extrude Cut**; **Type: Through All**
- Click **OK** and label *Mounting Hole*

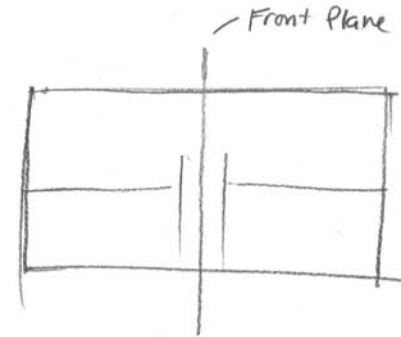


Solidworks Practice

Housing



Mirror Features



- Holding down CTRL key, highlight Front Plane (or plane 1) and *Mounting Hole* in the **Feature Manager** tree.
- Select **Insert, Pattern/Mirror, Mirror Feature**; All your selections should be in the proper selection boxes.
- Confirm this, and click **OK**. Leave this named *Mirror1*

Solidworks Practice

Housing



Add a Fillet

- Select the edge on the inside elbow of the base
- Select **Fillet**
- Input a Radius of **7.5mm**, press OK; leave this named *Fillet1*

