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 Though 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