



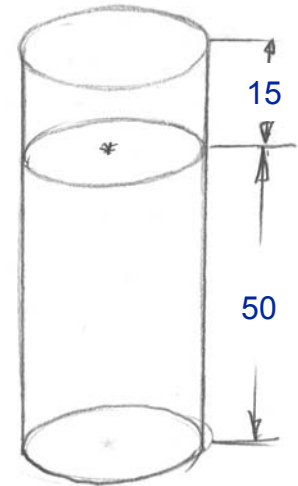
Solidworks Exercise

Modeling the piston (30 minutes)

- **Open** a new part file and **Save** as Piston.sldprt

Create the Cylinder

- Click on **Top Plane (or plane 2)** then open a sketch
- From the **origin**, click and drag a circle.
- **Dimension** the circle **20mm**
- Click the **isometric view** (if isometric icon is not visible go to view, toolbars, standard view)
- Hit the **Extrusion** key
- In Direction 1, type **15mm**, Blind
- In Direction 2, type **50mm**, Blind



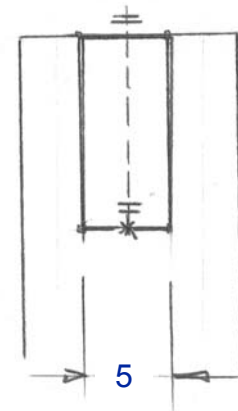
Solidworks Exercise

Modeling the piston (30 minutes)



Create a slot

- Click **Front View**
- Select **Front (or plane 1)** plane and open a sketch
- Select **centerline** tool, then draw a **centerline** vertically through the origin
- Select the line, then hit the **Mirror** tool
- Draw a half rectangle using the **line tool**
- Dimension the width to be **5mm**.
- Hit the **extruded cut** tool
- Select **Through All**, then check **Both Directions**
- Click OK and label *Slot*



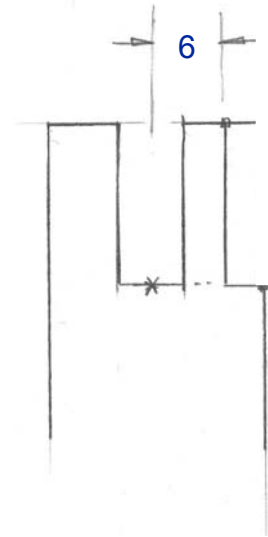
Solidworks Exercise

Modeling the piston (30 minutes)



Square off the ends

- Select **Front (or plane 1)** plane and open a sketch
- With **Line** tool, draw an L shape to the right of the origin. Start by snapping to the top edge.
- Select the bottom segment and while holding the **Cntrl** key down select the edge at the bottom of slot
- Right click the mouse and select ... **Add Relation**
- Click on '**Collinear**' and close
- Dimension the vertical line **6mm** to the right of the origin
- Select the **Cut** tool
- Select **Through All**; check **Both Directions**.
- Note arrow showing what will be cut away, then hit **OK**; Label *Flat face*



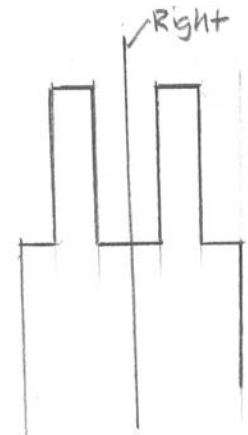
Solidworks Exercise

Modeling the piston (30 minutes)



Mirror squared end

- Select **insert, pattern/mirror, mirror feature**.
- Highlight **Mirror Plane** box, then select **Right (or plane 3)** plane in the **Features Manager Tree**
- Highlight **Features to mirror** box, then select *Flat face* in the **Features Manager Tree**, then hit **OK**
- Leave this labeled *Mirror1*



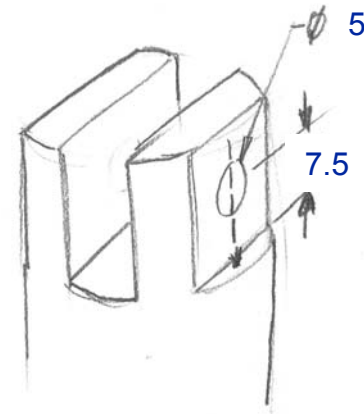
Solidworks Exercise

Modeling the piston (30 minutes)



Make a hole

- Select a face on the right 'squared end' plane
- Open a **sketch**
- Draw a **construction line** from Origin, upwards
- Select **Circle** tool and place the cursor on the construction line, then drag to create any sized circle
- Dimension circle to be **5mm** diameter
- Dimension distance of circle from origin to be **7.5mm**
- Click extrude cut, up to surface
- Click OK and label *Pin Hole*



Solidworks Exercise

Modeling the piston (30 minutes)



Chamfer the bottom

- Turn on **wireframe** viewing
- Select the bottom edge
- Hit **Chamfer** button
- Choose **Angle-Distance**, 1.5mm, 45 degrees

