SCHOOL OF MECHANICAL ENGINEERING

BMEE306P- CADFEA Lab

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| Tutorial	Topics
1	3D modelling using advanced curves and solid modelling commands

Objectives:

- Usage of advanced CAD commands to model plastic components

- Selection of appropriate materials (Metals and Non-metals)

- Create drafting for the 3D models according to the manufacturing process

Tutorial :1 : Modelling of a plastic knob

This lesson describes how to use different types of fillets.

In this lesson, you modify this knob part by:

- Adding different fillet types:
  - Face
  - Constant radius
  - Variable radius
- Using mirroring to assure symmetry
- Applying a library feature
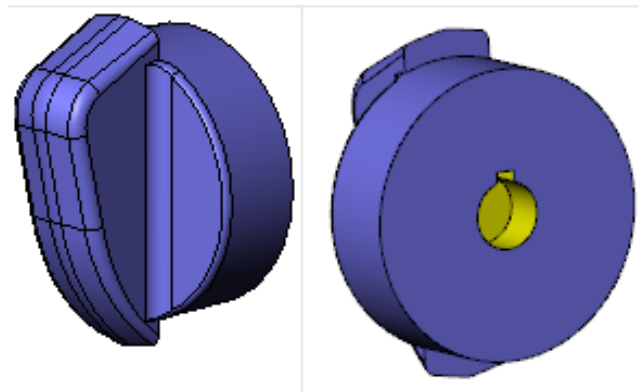
Notes :

- the base part can be got from C:\Users\Public\Documents\SOLIDWORKS\SOLIDWORKS 2021\samples\tutorial\fillets (based on the solidworks version pls change accordingly)
- all these tutorials are available in Solidworks help

In this section, you blend some of the faces using a face fillet with a hold line. For a face fillet, you can specify the radius between faces, or you can specify a hold line. When you specify a hold line, the face that shares an edge with the hold line is removed. The radius of the fillet is determined by the position of the hold line relative to the selected faces.

1. Do one of the following:

   a. Click here to open `Knob.sldprt` or browse to *drive letter:*`\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS` *version*`\samples\tutorial\fillets\knob.sldprt`.

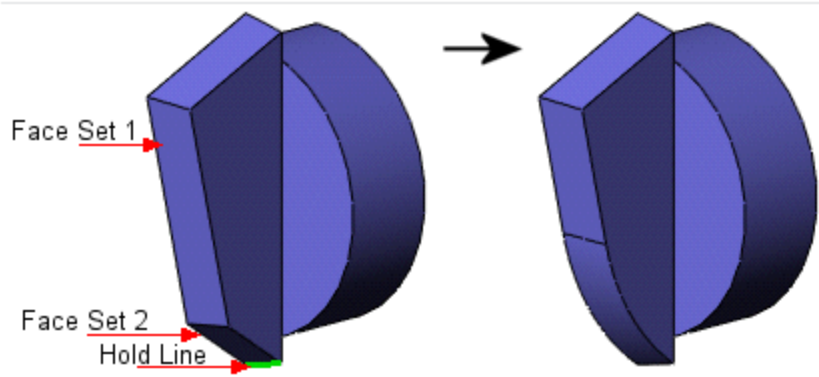   b. Build the sample part. Click here to learn how.

2. Click **Fillet** on the Features toolbar.

3. In the PropertyManager, under **Fillet Type** click **Face Fillet** .

4. Under **Items To Fillet**:

   a. Click inside the first selection box, **Face Set 1** . In the graphics area, select the face labeled below as Face Set 1.

   b. Click inside the second selection box, **Face Set 2** . In the graphics area, select the face labeled below as Face Set 2.

5. Under **Fillet Parameters**, for **Fillet Method**, select **Hold Line**. Click inside the selection box, **Hold Line Edges**, and select the edge labeled Hold Line below.
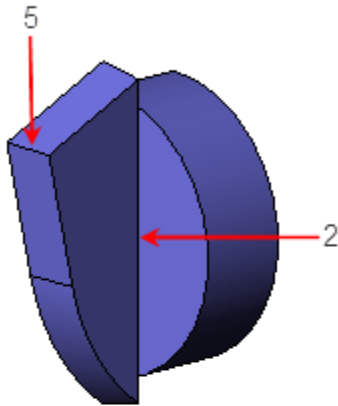

6. Click , then save the part.

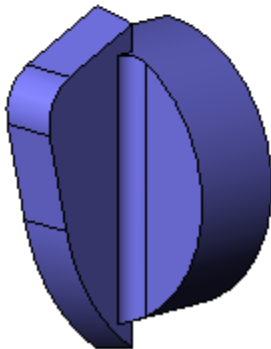# Creating Constant Size Fillets

Now you round some of the edges using constant size fillets.

1. Click **Fillet** ▣ on the Features toolbar.

2. In the PropertyManager, under **Fillet Type** click **Constant Size Fillet** ▣ .

3. Click the edge of the grip labeled 5.

4. Under **Items To Fillet**, select **Full preview**.
   A preview of the fillet appears in the graphics area.

5. Under **Fillet Parameters** set **Radius** ⌐ to 5.

6. Click ✔ .

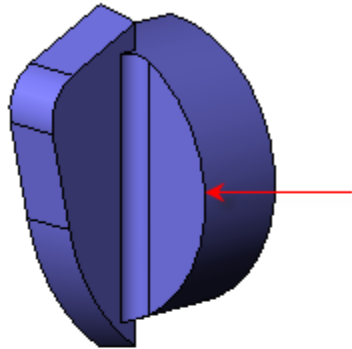7. Repeat steps 1 through 6 to add a fillet to the edge labeled 2. Change the radius values to 2.

   When filleted edges intersect, it is good practice to add the larger fillet first.

Creating an Asymmetric Fillet

Next you create an asymmetric fillet along the cylindrical edge.

1. Click the edge shown and click **Fillet** [icon] on the Features toolbar.

2. In the PropertyManager, click **Constant Size Fillet** [icon].

3. Under **Fillet Parameters**, for **Fillet Method**, select **Asymmetric**.

4. For **Distance 1** [icon], type 1.

5. For **Distance 2** [icon], type 2. If necessary, click **Reverse Direction** [icon] to assign the larger radius to the outer side of the fillet.
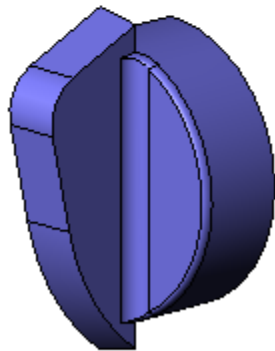
6. Click [checkmark].
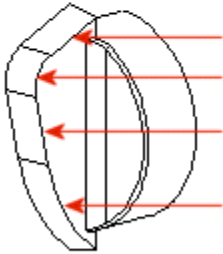
Creating a Variable Size Fillet

You create variable radius fillets by specifying a different radius for each vertex of the edges you want to fillet.

1. Click **Hidden Lines Removed** ▢ on the View toolbar.

2. Click **Fillet** ▢ on the Features toolbar.

3. In the PropertyManager, under **Fillet Type**, click **Variable Size Fillet** ▢.

4. Under **Items to Fillet**, clear **Tangent propagation**.

5. For **Edges to Fillet**, select the edges shown.

6. Under **Variable Radius Parameters**, for **Fillet Method**, select **Symmetric**.

Completing the Variable Size Fillet

Under **Variable Radius Parameters**, set the radius values for the five vertices as shown in the illustration.

1. Under **Variable Radius Parameters**, in **Radius** ↖, type 1.5 and click **Set All**.

   The value for each vertex in **Attached Radii** 🔄 changes to 1.5.

2. In **Attached Radii** 🔄, Ctrl+ select **V1** and **V5**.

3. In **Radius** ↖, type 1.

   In **Attached Radii** 🔄, the radii for vertices **V1** and **V5** change to 1. The fillet previews appear in the graphics area.
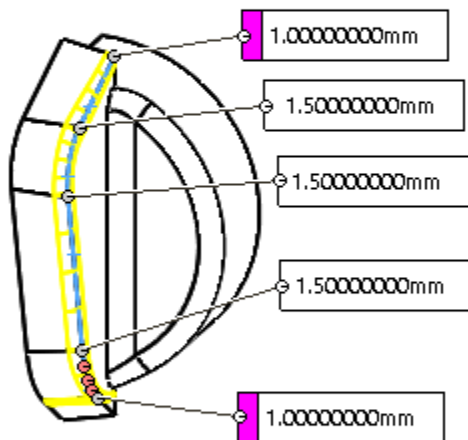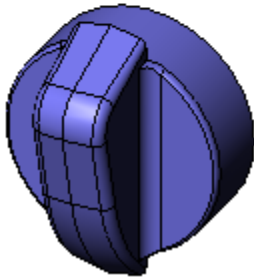

4. Click ✔.

   To verify the radius values, double-click **VarFillet1** in the FeatureManager design tree. The values appear in the graphics area. Click anywhere in the graphics area to hide the values.

5. Click **Shaded With Edges** 🔲 on the View toolbar, and save the part.

Mirroring the Model

To take advantage of the part's symmetry and to finish the part, mirror the part about the **Right** plane.

1. Click **Trimetric** on the Standard Views toolbar.

2. Click **Mirror** on the Features toolbar.

3. In the PropertyManager, for **Mirror Face/Plane**, select the **Right** plane from the flyout FeatureManager design tree.

4. Under **Bodies to Mirror**, click any geometry in the graphics area.

    A preview of the mirrored model appears in the graphics area.

5. Click ✔.

    A mirror image of the original part is joined to the part at the selected plane to make a complete, symmetrical part.

Filleting the Parting Line

When you mirrored the drafted grip, it created a parting line along the top of the grip. Smooth the parting line by adding a constant radius fillet.

1. Click **Dimetric** on the Standard Views toolbar.

2. Click **Fillet** on the Features toolbar and select the edge shown.
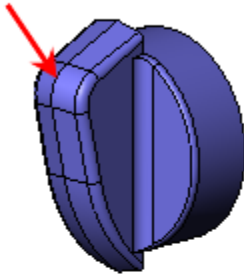
3. Under **Fillet Type**, click **Constant Size Fillet** .

4. Under **Fillet Parameters**, set **Radius** to 5.

5. Make sure **Tangent propagation** is selected, so that the fillet extends along all of the segments of the edge.

6. Click .
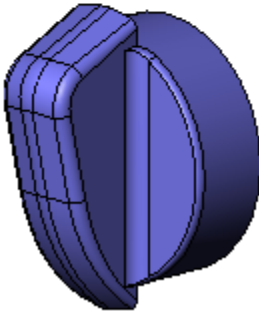
Inserting a Library Feature

A library feature is a frequently used feature, or combination of features, that you create once and then save in a library for future use. The SOLIDWORKS software comes with several pre-made library features in the Design Library. Here, you insert a keyway.

1. Click **Back** 🔲 (Standard Views toolbar).

2. Select the Design Library 📦 tab on the Task Pane.

3. Expand Design Library 📦.

   If you are missing the Design Library folder, you can add it manually by clicking **Add File Location** 📦 and browsing to `drive letter:\ProgramData\SOLIDWORKS\SOLIDWORKS version\design library` in the dialog box and then clicking **OK**.

4. Navigate to `features\metric\keyways`.

5. Drag `bore with square keyway bs 4500p1` onto the part.
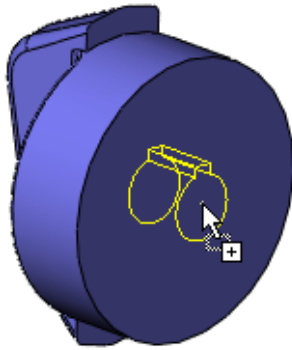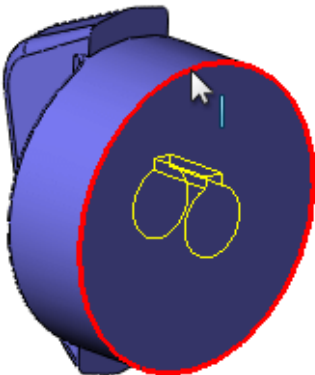

6. In the PropertyManager, under **Configuration**, select:

   - `2 X 1 ø    7`
   - **Link to library part**.

     If you edit the library feature, this option ensures that the changes you make in the original library feature are applied to this part.
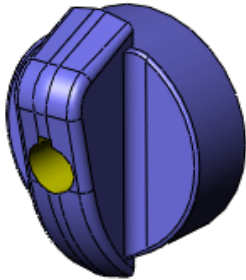
7. In the graphics area, select the outer edge for the reference edge, then click ✔.

Editing a Library Feature

When you edit a library feature, you edit it in the context of the `*.sldlfp` file, which is the file extension for library features.

1. Rotate the part. Notice the keyway cuts through the entire part.

2. In the FeatureManager design tree, right-click **bore with square keyway bs 4500p1** and select **Edit In Context**.

   The `*.sldlfp` file opens.

3. In the FeatureManager design tree, right-click **Keyway** and select **Edit Feature** .

4. In the PropertyManager, under **Direction 1**:
   a. Select **Blind** in **End Condition**.
   b. Set **Depth** to 5.

5. Click ✔ .

6. Click **Window** > **knob.sldprt** to return to the part window.

   The part updates with the modified keyway.
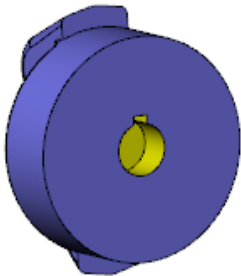
Expected Output : The drafting should have views (isometric,sectional and other relevant views) that can be used to design a die ; Show the mass of the knob with the probable plastic that can be used for this product. Please add comments on shrinkage allowance etc in the drafting.

**References:** https://www.nicoletplastics.com/resources/blog/8-factors-in-plastic-part-design-for-manufacturability/
https://www.rodongroup.com/blog/bid/93916/key-ingredients-to-achieving-perfect-plastic-parts
https://www.essentracomponents.com/en-gb/news/guides/a-guide-to-knobs-types-materials-and-applications

Tutorial -2 : Model a lamp stand using Revolve and Sweep Features

In this lesson, you create the candlestick shown below. This lesson demonstrates:

- Creating a revolve feature
- Creating a sweep feature
- Creating an extruded cut feature with a draft angle
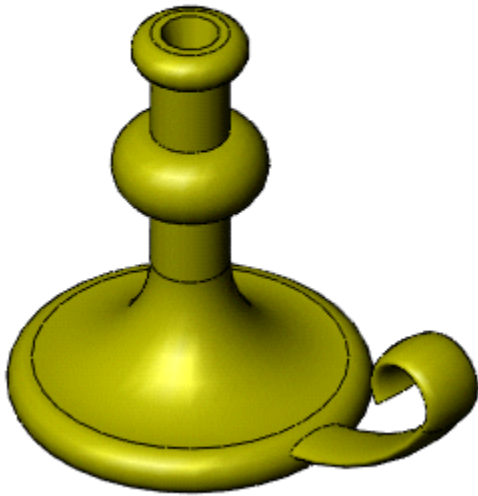
Sketching a Revolve Profile

You create the base feature of the candlestick by creating a sketch profile and revolving the sketch profile around a centerline.

1. Click **New** ⬜ on the Standard toolbar and create a new part.

2. Click **Revolved Boss/Base** 🌀 on the Features toolbar.
   The **Front, Top,** and **Right** planes appear.

3. Select the **Front** plane.
   A sketch opens on the **Front** plane.

4. Click **Line** ✏️ on the Sketch toolbar. Sketch a vertical line from the origin, and sketch the two horizontal lines as shown below.

5. Click **Smart Dimension** 🔧 on the Sketch toolbar. Dimension the sketch as shown.
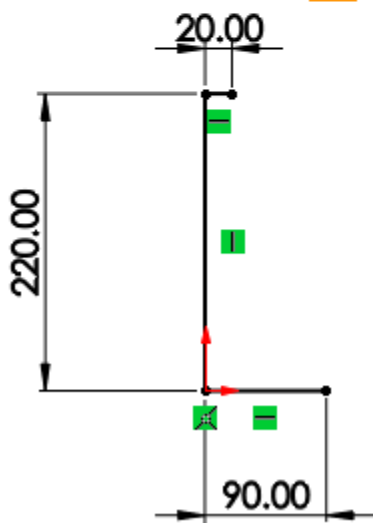
Sketching the First Revolve Profile Arc

Now sketch and dimension the arcs and lines needed to complete the profile. First create the small arc at the top of the sketch.
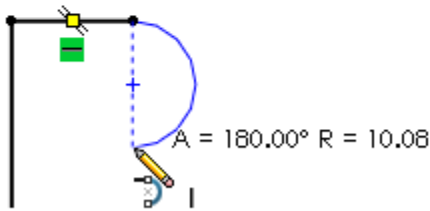
1. Click **Tangent Arc** ⬡ (Sketch toolbar).

   > You might need to click the down arrow on the **Arc** ⬡ ▾ flyout button to select the **Tangent Arc** tool.

2. Click the endpoint of the top horizontal line, move the pointer to the right, then downward.

   > Watch the pointer for feedback and for inferencing. As you sketch, inferencing pointers and lines help you align the pointer with existing sketch entities and model geometry.

3. When the radius is approximately 10mm (**R=10**) and the vertical inferencing line is visible, click again.

   A = 180.00° R = 10.08

4. Click **Smart Dimension** ⬡ on the Sketch toolbar and dimension the arc radius to 10.

Sketching the Second Revolve Profile Arc

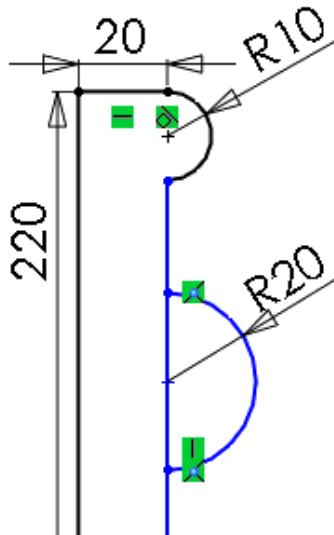Now create the vertical line and the second arc.

1. Click **Line** ✏️ on the Sketch toolbar.

2. Sketch a vertical line downward approximately 150mm long, starting at the lower endpoint of the arc.

   > Do not dimension the line at this time.

3. Click **3 Point Arc** 🔵 on the Sketch toolbar.

   > You might need to click the down arrow on the **Arc** flyout button to select the **3 Point Arc** tool.

4. Sketch an arc so that the arc centerpoint and endpoints are coincident with the li

   (Watch for the ✏️ pointer.) Use the following measurements:
   - Length approximately 40mm (**L=40**)
   - Angle approximately 180° (**A=180**)
   - Radius approximately 20mm (**R=20**)

5. After clicking to end the arc, set the angle to 180° in the **Parameters** section of t PropertyManager.

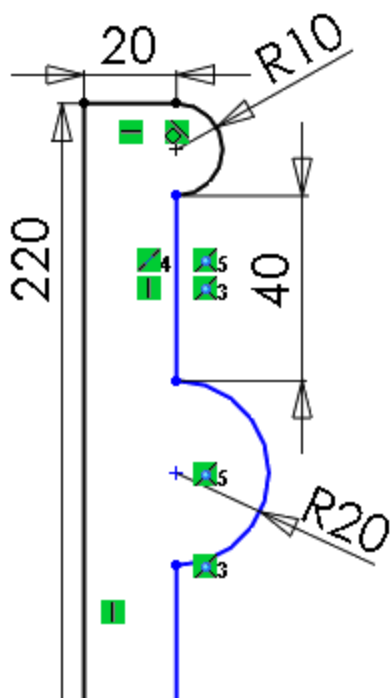6. Click **Smart Dimension** 📐 on the Sketch toolbar, and then dimension the arc ra to 20.

7. Click **Trim Entities** ✂️ on the Sketch toolbar.

8. In the PropertyManager, under **Options**, click **Trim to closest** ⊥ .

**9.** Select the highlighted segment to delete it.
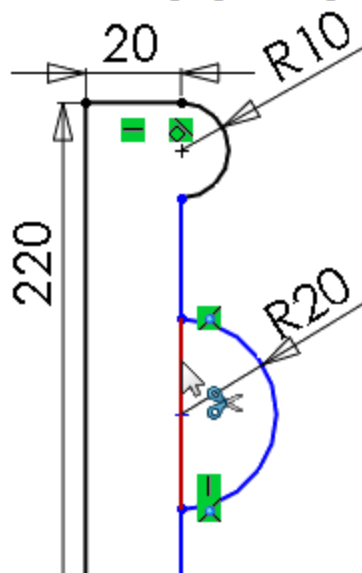
0. Right-click in the graphics area and select **Smart Dimension** . Dimension the upper vertical line to 40, as shown.

Completing the Revolve Profile

Now add relations and a tangent arc.

1. Click **Select** ⬚ on the Standard toolbar, then hold down **Ctrl** and select the vertical lines on each side of the lower arc.

2. In the PropertyManager, under **Add Relations**, click **Equal** =, then click **OK** ✓.
   The **Equal** relation ensures that both vertical lines will maintain equal length.

3. Click **Tangent Arc** ⬚ on the Sketch toolbar, then click the endpoint of the lower vertical line.

4. Move the pointer downward to create an arc that has an angle of 90° and a radius of approximately 60mm. Click to place the arc.
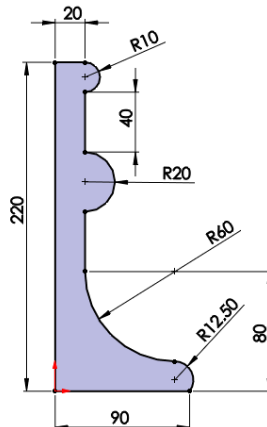
A = 90.60° R = 58.33

90.00
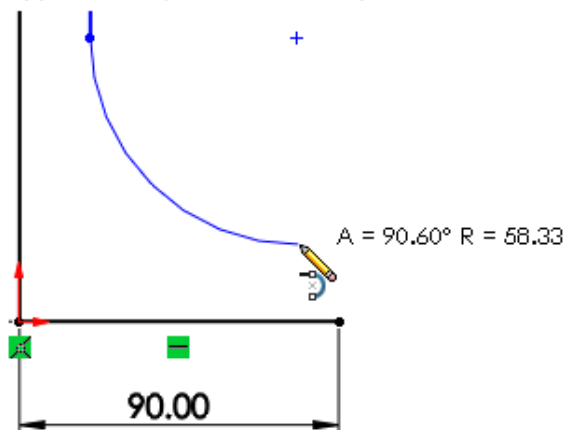
5. Sketch another tangent arc. Move the pointer until the endpoint of the arc is coincident with the endpoint of the bottom horizontal line as shown.

   📹 Video: Sketching the Tangent Arc
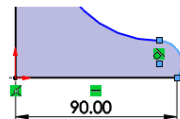
90.00

6. Click **View > Hide/Show > Sketch Relations** to hide the sketch relations in the graphics area.

7. Dimension the rest of the sketch as shown.

20
R10
40
R20
R60
R12.50
80
220
90

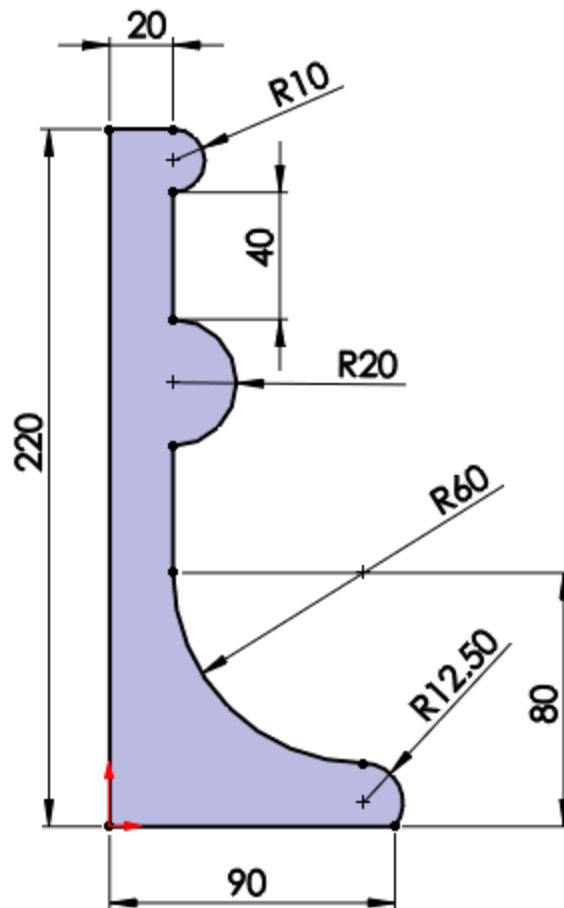When you are done dimensioning, the sketch is fully defined (All lines and endpoints are black).

6. Click **View** > **Hide/Show** > **Sketch Relations** to hide the sketch relations in the graphics area.

7. Dimension the rest of the sketch as shown.

When you are done dimensioning, the sketch is fully defined (All lines and endpoints are black).

Creating the Revolve Feature

Now that you created the sketch profile, you revolve the profile around the centerline to create the revolve feature.

1. Click **Exit Sketch** ⤵ on the Sketch toolbar.

   The Revolve PropertyManager appears.

   > If you move the pointer over a box or an icon in the PropertyManager, a tooltip appears with the name of the box or icon.

2. For **Axis of Revolution** ⸝, select the long vertical line in the sketch.

3. Under **Direction1**:

   a. In **Revolve Type**, select **Blind**.

   b. Set **Direction 1 Angle** ⤶ to 360.

   c. Click ✔.

   The Revolve feature is created.
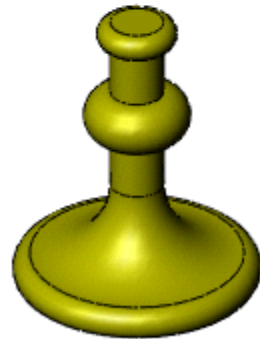
4. Save the part as `Cstick.sldprt`.
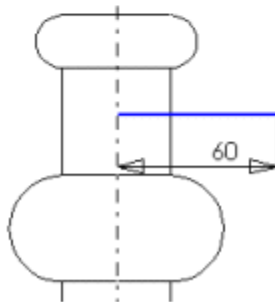
Sketching the Sweep Path

A sweep is a base, boss, or cut created by moving a section along a path. In this part of the tutorial, you create the candlestick handle by using a sweep. First, you sketch the sweep path.

The path can be an open curve, or a closed, non-intersecting curve. Neither the path nor the resulting sweep can be self-intersecting.

1. Select the **Front** plane in the FeatureManager design tree, then click **Sketch** on the Sketch toolbar to open a new sketch.

2. Click **Front** on the Standard Views toolbar.

3. Click **Hidden Lines Removed** on the View toolbar.

4. Click **View > Hide/Show > Temporary Axes**.

   Notice that the temporary axis of the revolved base appears.

5. Right-click in the graphics area and select **Sketch Entities > Line**, then move the pointer over the temporary axis.

   The pointer changes to indicating that the pointer is exactly on the temporary axis.

6. Sketch a horizontal line as shown, and dimension the line to 60.

Completing the Sweep Path

1. Right-click in the graphics area and select **Recent Commands > Tangent Arc.**

2. Sketch an arc starting at the endpoint of the line.

3. Dimension the arc to a radius of 150.

    > If the radial dimension is out of view, click the Leaders tab in the Dimension
    > PropertyManager. Click **Foreshorten** ⇜ , then click ✔ .

4. Select the endpoints of the arc and set the vertical dimension to 65.

    > As you move the pointer, the dimension snaps to the closest orientation. When the
    > preview indicates the dimension type and location you want, right-click to lock the
    > dimension type. Click to place the dimension.
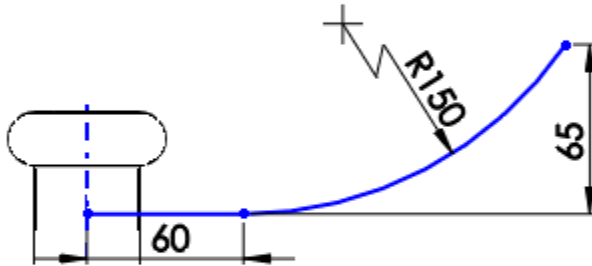
5. Right-click and select **Recent Commands > Tangent Arc**, then sketch another arc as shown.

6. Dimension it to a radius of 20.
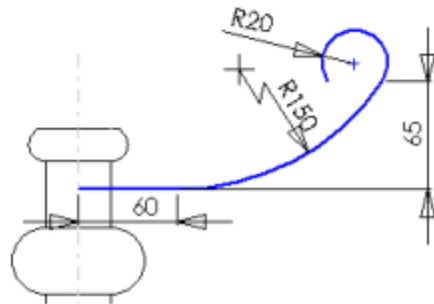
Adding Relations to the Path

Now add relations to control the sweep path.

1. Click **Select** [cursor icon] on the Standard toolbar, then hold down **Ctrl** and select the endpoints of the tangent arc you just sketched.
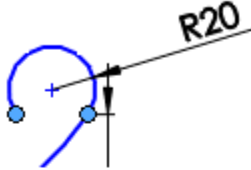
   The Properties PropertyManager appears. The two endpoints are listed under **Selected Entities**.

2. Under **Add Relations**, click **Horizontal** ▬.

3. Click ✔.

   The dimensions and relations prevent the sweep path from changing size and shape when moved.

4. Click **Display/Delete Relations** [icon] on the Sketch toolbar.

   The Sketch Relations PropertyManager lists all the relations in the current sketch, including both relations that are added automatically as you sketch and relations that you add manually. For example, the coincident relation between the sweep path and the revolved base was added automatically. You control the type of relation you want to see with the **Filter** option.

5. In the PropertyManager, under **Relations**, select **All in this sketch** in **Filter**.

6. Select each relation in **Relations**.

   As you select each relation, its entities are highlighted in the graphics area.

7. Click ✔.

   About Sketch Relations

Dimensioning the Sweep Path

Next, dimension the sweep path with respect to the revolved base.

1. Dimension the distance between the horizontal line of the sweep path and the bottom edge of the revolved feature to 10.
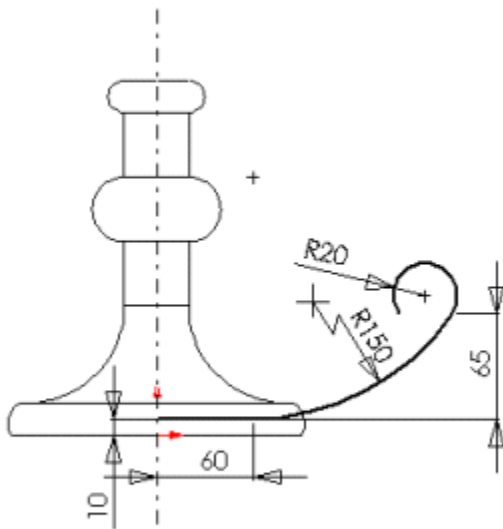
The sweep path is fully defined.

2. Click **Exit Sketch** ⤶ on the Sketch toolbar.

Sketching the Sweep Section

After you sketch the sweep path, you need to sketch the sweep section.

1. Select the **Right** plane in the FeatureManager design tree, then click **Sketch** ⊞ on the Sketch toolbar to open a new sketch.

2. Click **Normal To** ⬍ on the Standard Views toolbar.

3. Click **Ellipse** ⊘ on the Sketch toolbar, then sketch an ellipse anywhere in the graphics area.

    > To sketch an ellipse, drag horizontally from the center point of the ellipse to set the width of the ellipse, release the pointer, then click and drag vertically to set the height.

4. Click **Select** ▷ on the Standard toolbar, then hold down Ctrl and click the endpoints of the ellipse as shown.
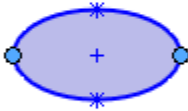
5. In the PropertyManager, under **Add Relations**, click **Horizontal** ▬ , then click **OK** ✔ . This relation ensures that the ellipse is not slanted.

6. Dimension the ellipse as shown.
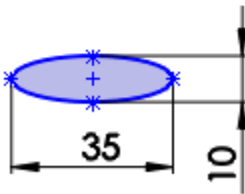
7. Click **OK** ✔ .

Completing the Sweep Section Sketch

1. Click **Isometric** on the Standard Views toolbar.

2. Hold down **Ctrl** and click the center point of the ellipse and the endpoint of the horizontal line of the sweep path.
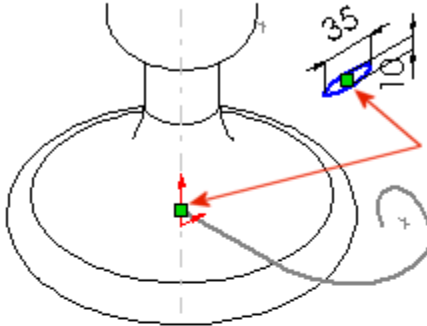
3. In the PropertyManager, under **Add Relations**, click **Coincident** , then click **OK** .
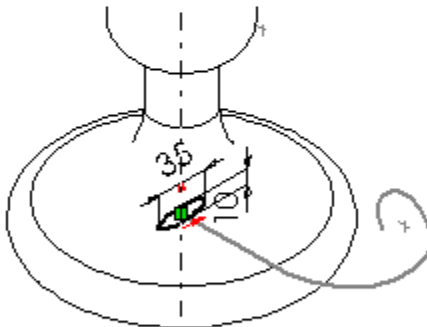
4. Click **View** > **Hide/Show** > **Temporary Axes** to hide the temporary axis.

5. Click **Exit Sketch** on the Sketch toolbar.

Creating the Sweep

Now you combine the sweep path and sweep section sketches to create the sweep.

1. Click **Swept Boss/Base** 🪱 on the Features toolbar.

2. In the PropertyManager:

   a. Select **Sketch3** (the ellipse) in the graphics area for **Profile** C^0.

   b. Select **Sketch2** (the path) in the graphics area for **Path** C.
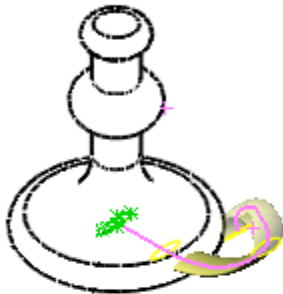
   A preview of the sweep appears in the graphics area. Note how the colors in Profile and Path match those in the graphics area.

3. Under **Options**, select **Follow Path** in **Profile orientation**.

4. Click **OK** ✔ to create the sweep.

5. Click **Shaded with Edges** 🧊 (View toolbar).

   The candlestick's handle is complete.

6. Save the part.

Completing the Part

The final step is to create a cut to hold a candle.

1. Select the top face of the revolved base feature, then click **Extruded Cut** ⬚ on the Features toolbar.

2. Click **Normal To** ⬚ on the Standard Views toolbar.

3. Click **Circle** ⬚ on the Sketch toolbar, and select the sketch origin. Sketch and dimension a circle as shown.
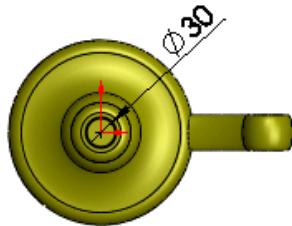

4. Click **Exit Sketch** ⬚ on the Sketch toolbar.
   The Cut-Extrude PropertyManager opens.

5. Click **Isometric** ⬚ (Standard Views toolbar).

6. In the PropertyManager, under **Direction 1**:
   a. Select **Blind** in **End Condition**.

   > Click **Reverse Direction** ⬚ if necessary to make the arrow point down.
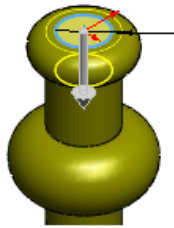
   b. Set **Depth** ⬚ to 25.
   c. Click **Draft On/Off** ⬚, and set **Draft Angle** to 15.

7. Click ✔.
   The cut is added to the top of the candlestick.

Viewing the Cut

You can make the part transparent to help to see the angled cut.

1. At the top of the FeatureManager design tree, to the right of the tabs, click **Show Display Pane** ❯.

2. Move the pointer over **cstick** at the top of the FeatureManager design tree, and then across into the **Transparency** 🔷 column.
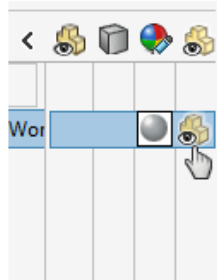
3. When the pointer changes to 👆 , click in the column.

   In the graphics area, the part becomes transparent. You can see the angled cut in the top of the candlestick.
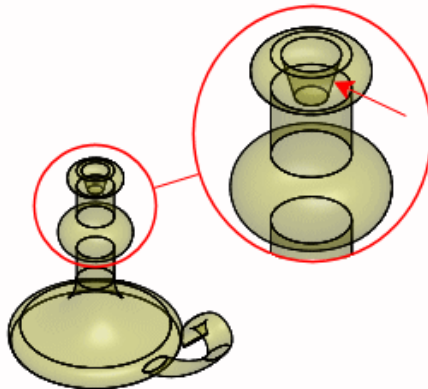
4. Click again in the **Transparency** 🔷 column to return the part to its original appearance.

5. Click **Hide Display Pane** ❮.

Expected Output : **The drafting should have views that can be used to manufacturing of the lampstand ; Show the mass of the with choice of the material be used for this product**

Tutorial : 3 :Modelling a hammer head using Loft Features

In this lesson, you create this hammer head using loft features.

A loft is a base, boss, or cut created by connecting multiple cross sections, or profiles.

This lesson demonstrates the following:

- Creating planes
- Sketching, copying, and pasting the profiles
- Creating a solid by connecting the profiles (lofting)
- Adding a flex feature to bend the model
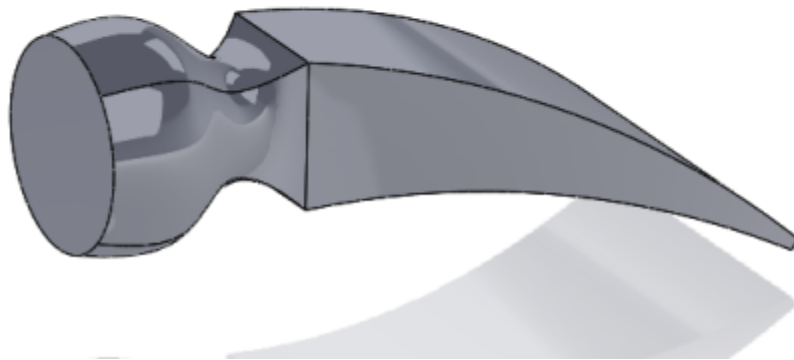
Setting Up the Planes

To create a loft, you begin by sketching the profiles on faces or planes. You can use existing faces and planes, or create new planes.

1. Click **New** ⬜ on the Standard toolbar and create a new part.

   The planes in a SOLIDWORKS model are not always visible. However, you can display them. For this lesson, it is helpful to display the **Front** plane.

2. Click **View> Hide/Show** and verify that **Planes** is selected.

3. Right-click the **Front** plane in the FeatureManager design tree and select **Show** 👁.

   The **Front** plane appears in the graphics area.

4. With the **Front** plane still selected, click **Plane** ⬚ on the Reference Geometry toolbar.

   The Plane PropertyManager appears. A preview of the new plane appears in the graphics area. Under **First Reference**, **Front** is listed in the **First Reference** ⬚ box.

5. Set **Offset distance** ⬚ to 25 and click ✔.

   A new plane, **Plane1**, is created in front of the **Front** plane.

The planes used in a loft do not have to be parallel, but they are for this lesson.

Completing the Planes

1. With **Plane1** selected, click **Plane** ⬚ on the Reference Geometry toolbar again, and add another offset plane at a distance of 25mm. (This is **Plane2**).

2. Click **OK** ✓ .

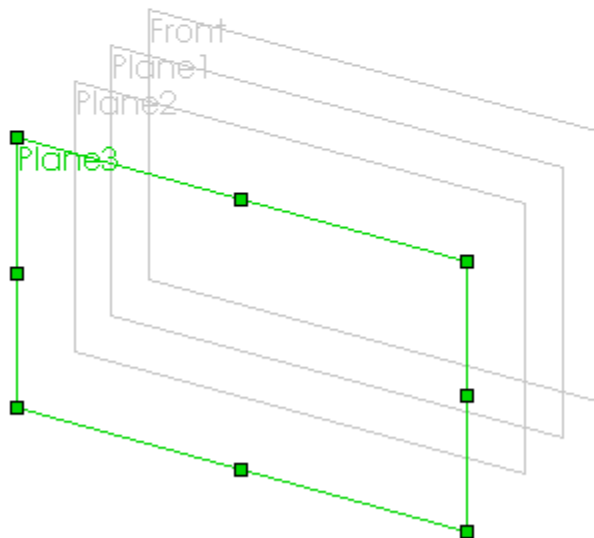   > Another way to create an offset plane is to copy an existing plane.

3. Hold down **Ctrl**, select an edge of **Plane2** in the graphics area, and drag to a location in front of **Plane2**.

   Another offset plane, **Plane3**, is created.

4. To set the offset distance for the new plane, set **Offset distance** ⬔ to 40 in the PropertyManager.

5. Click **OK** ✓ .

   Your graphics area should look like this image.

Sketching the Profiles

You create the body of the hammer head by lofting between simple profile sketches.

1. Select the **Front** plane, then click **Sketch** ⊞ on the Sketch toolbar.

   The view orientation changes to a front view.

   > You may find it easier to see the sketch entities if the planes are not shaded. Click
   >
   > **Options** ⚙ on the Standard toolbar. On the System Options tab, select **Display**.
   > Clear **Display shaded planes** and click **OK**.

2. Sketch and dimension a 60mm square as shown to center it about the origin.
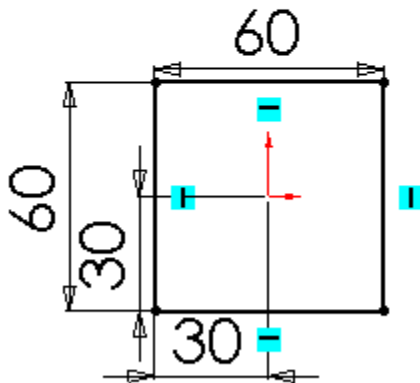
3. Exit the sketch.

Completing the Profiles

1. Open a sketch on **Plane1**, and sketch a circle, centered on the origin.

   It appears as though you are sketching on top of the first sketch. However, the first sketch is on the **Front** plane, and it is not affected by sketching on **Plane1**, a parallel plane in front of it.
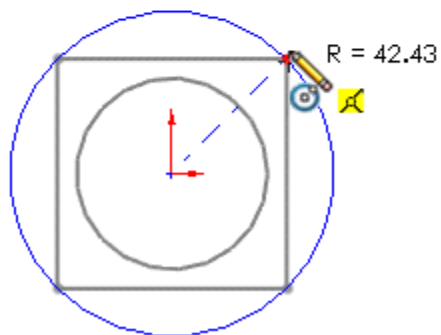
2. Dimension the circle to 50mm in diameter.
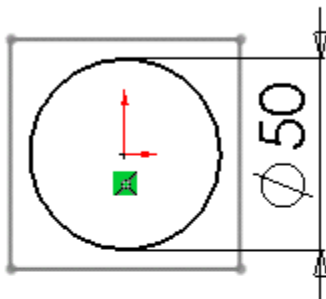
3. Exit the sketch.

4. Open a sketch on **Plane2**, and sketch a circle, centered on the origin. As you drag, make the diameter of the circle coincident with the vertex of the square. (Watch for the

    pointer.)

5. Exit the sketch.

Copying a Sketch

You can copy a sketch from one plane to another to create another profile.

1. Click **Isometric** on the Standard Views toolbar.
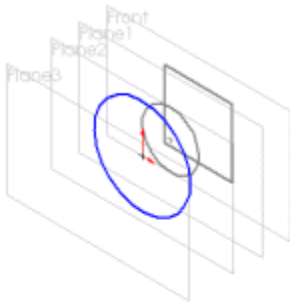
> If a sketch is on the wrong plane, you can change the plane. Right-click the sketch in the FeatureManager design tree or the graphics area, and select **Edit Sketch Plane** . Select the new plane for the sketch, then click **OK** ✔ in the Sketch Plane PropertyManager.

2. Select **Sketch3** (the larger circle).

3. Click **Copy** on the Standard toolbar.

4. Select **Plane3**.

5. Click **Paste** on the Standard toolbar.

Creating the Loft

Now use the **Lofted Boss/Base** feature to create a solid model based on the profiles.

1. Click **Lofted Boss/Base** ⬇ on the Features toolbar.

2. In the graphics area, click near the same place on each profile (for example, the upper-right side), so the loft path travels in a straight line and does not get twisted. Select the sketches in the order you want to connect them.
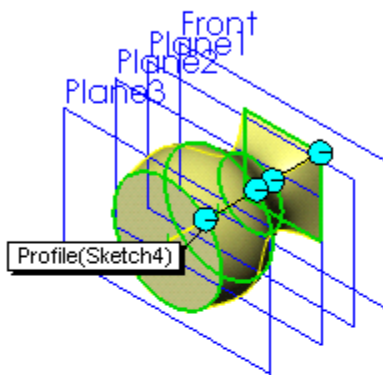
A preview shows you how the profiles will be connected. The system connects the points or vertices closest to where you click.

3. Examine the preview of how the profiles will be connected.
   - If the sketches appear to be connected in the wrong order, you can use the **Move Up** ⬆ or **Move Down** ⬇ buttons under **Profiles** in the PropertyManager to rearrange the order.
   - If the preview indicates that the wrong points will be connected, right-click in the graphics area, select **Clear Selections**, and select the profiles again.

4. Click ✔ to create the solid model.
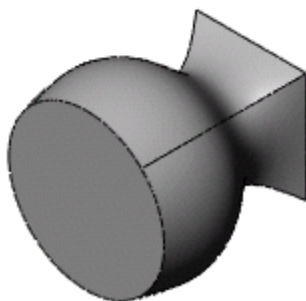
Creating a Boss Loft

For the pointed end of the hammer head, you create another loft.

1. Hold down **Ctrl**, and drag an edge of the **Front** plane to create an offset plane behind the original **Front** plane.

   The Plane PropertyManager appears.

2. Set **Offset distance** to 200 .

3. Make sure that **Flip offset** is selected so the new plane is created behind the **Front** plane, then click **OK** ✓ to create the new **Plane4**.

4. Click **Hidden Lines Removed** on the View toolbar.

5. Click **Normal To** on the Standard Views toolbar.

6. Open a sketch on **Plane4**, then sketch and dimension a narrow rectangle as shown, which is the profile you use to create the next loft.
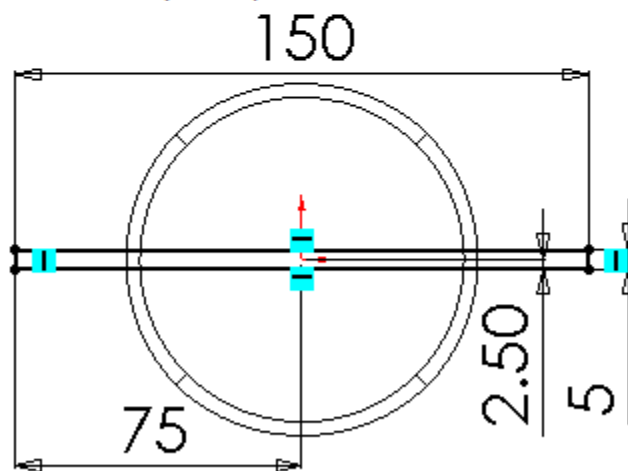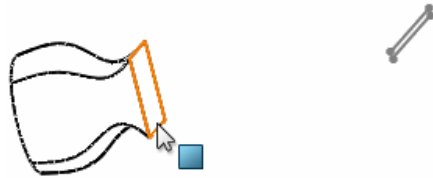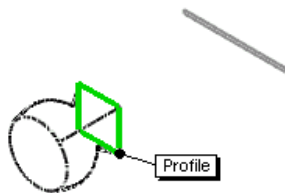

7. Exit the sketch.

Completing the Second Lofted Boss

Now you complete the second lofted boss.

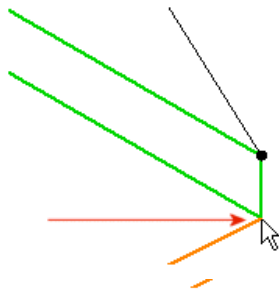1. Click **Isometric** on the Standard Views toolbar.

2. Click **Lofted Boss/Base** on the Features toolbar.

3. Select the square profile:

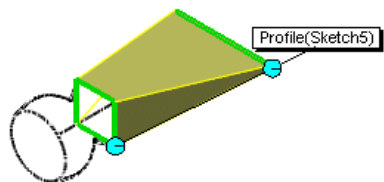   a. Rotate the model as shown and select the face in the lower corner closest to you.

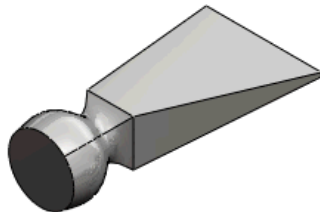   b. Click **Isometric** again.

4. Click the lower part of the narrow rectangular sketch.

5. Examine the preview of how the two profiles will be connected.

6. Click ✔.

7. Click **Shaded With Edges** on the View toolbar, and save the part.


Bending the Part with the Flex Feature

The flex feature deforms a model. You can use the flex feature to bend, twist, taper, or stretch a model. Here you use the flex feature to bend the hammer head.

1. Click **Flex** [icon] on the Features toolbar.

2. In the PropertyManager, under **Flex Input**, select:

   a. The part in the graphics area for **Bodies for Flex** [icon].
   b. **Bending**.

3. Under **Trim Plane 2**, click in **Select a reference entity for Trim Plane 2** [icon].
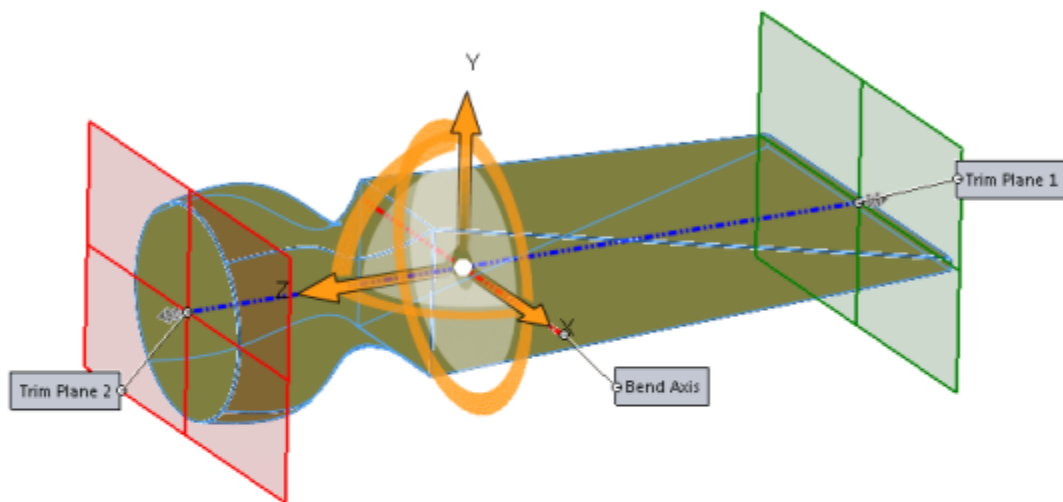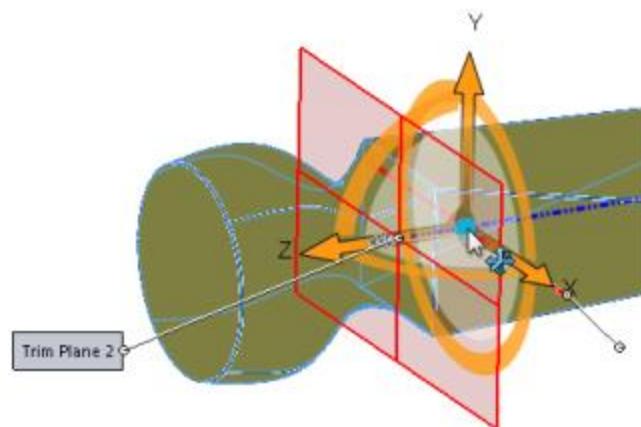

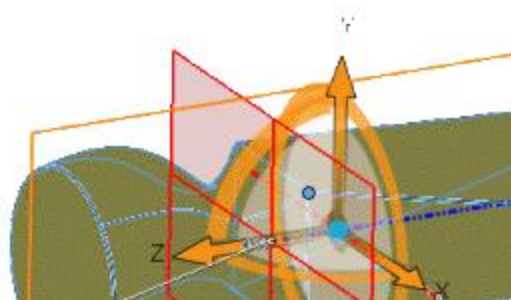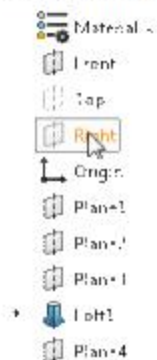4. In the graphics area, select the vertex as shown.
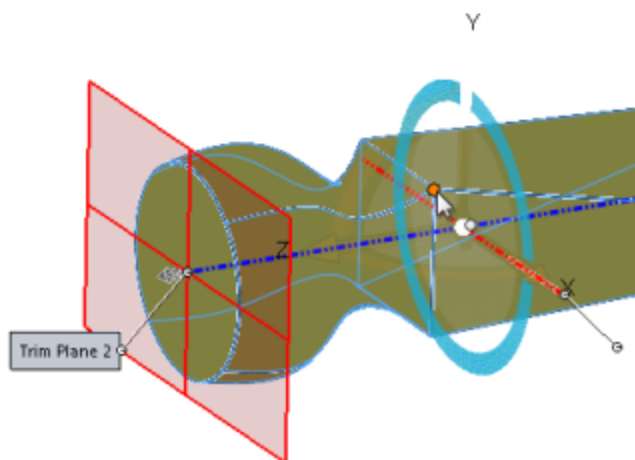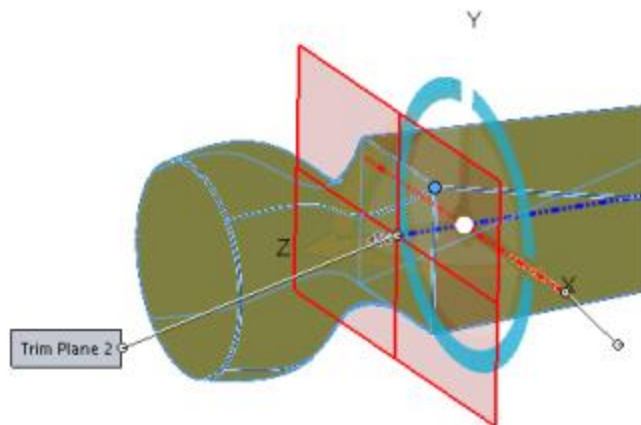
5. Right-click the triad's center sphere as shown, and select **Align to**.


If you do not see this option, click in **Bodies for Flex** and try again.

6. Expand the flyout FeatureManager design tree and select the **Right** plane to align the trim plane axis (Z axis on the triad) parallel to the **Right** plane.
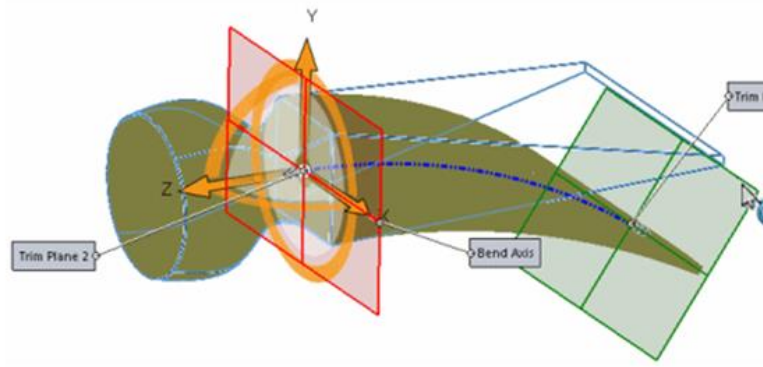
Completing the Bend

1. Right-click the triad's center sphere and select **Move Triad to Plane 2**.
   The center of the model is aligned with the center of the triad.
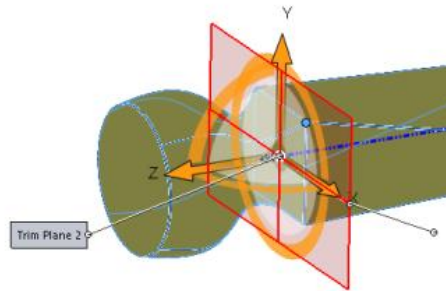
2. Drag the pointer over an edge of **Trim Plane 1**. When the pointer changes to ,
   click and drag the pointer up and down.
   Only the material between the trim planes moves.

3. Click ✔.

Expected Output : **The drafting should have views that can be used to manufacture the hammer head ; Show the mass of the hammer head with high strength steel that can be used for this product**

Sample drafting images: