

SolidWorks Part 2

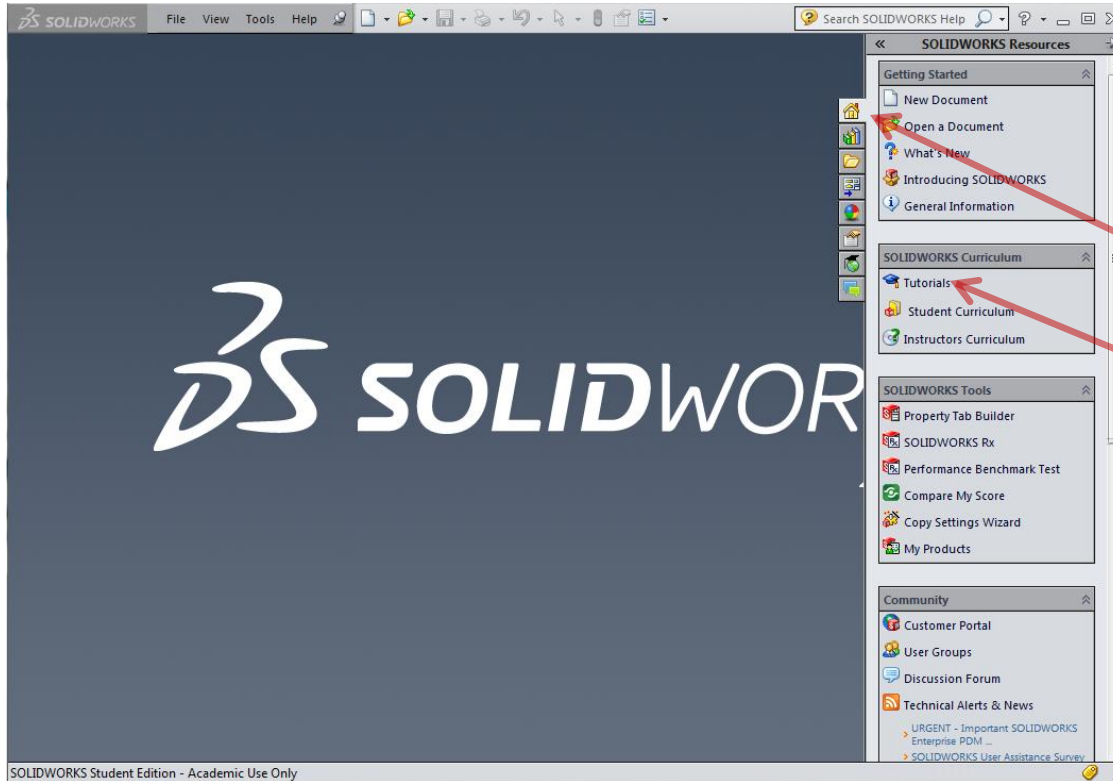
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Introductory Medical Device Prototyping

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Starting The Tutorials



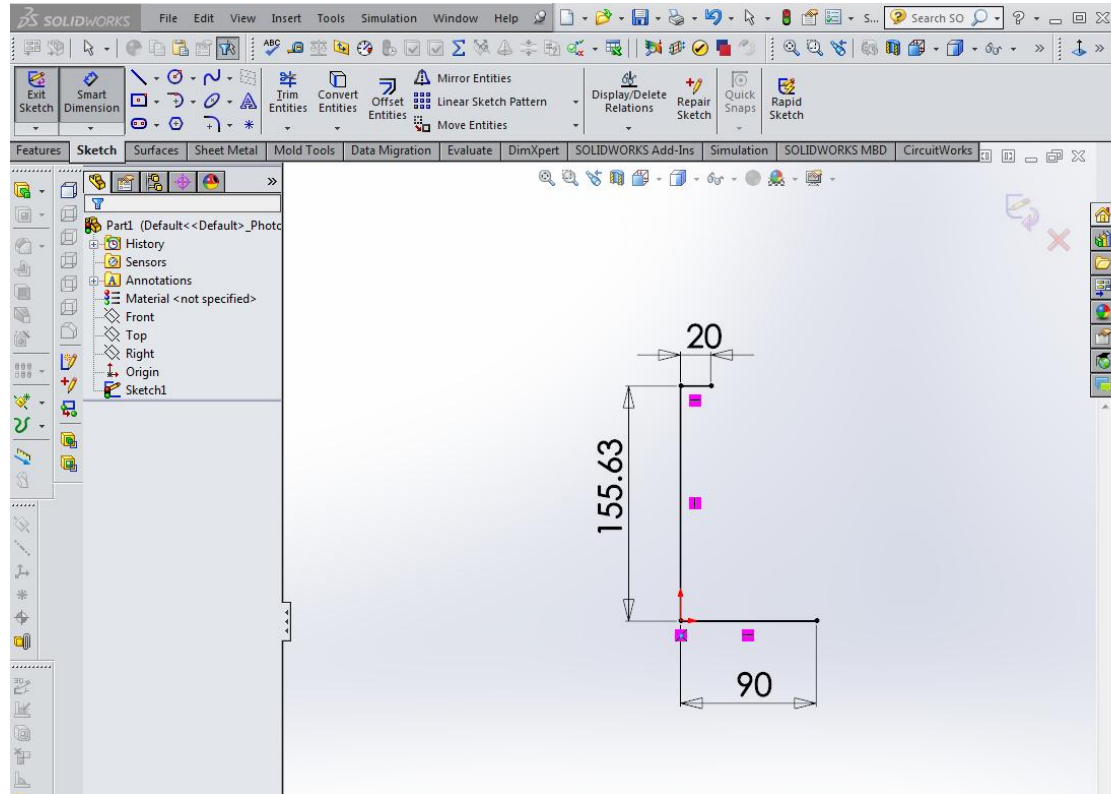
- Launch SolidWorks
- Select Resources to open the Task Pane.
- Select Tutorials

Select Basic Techniques

Open Revolves and Sweeps



Sketch the Revolve Profile

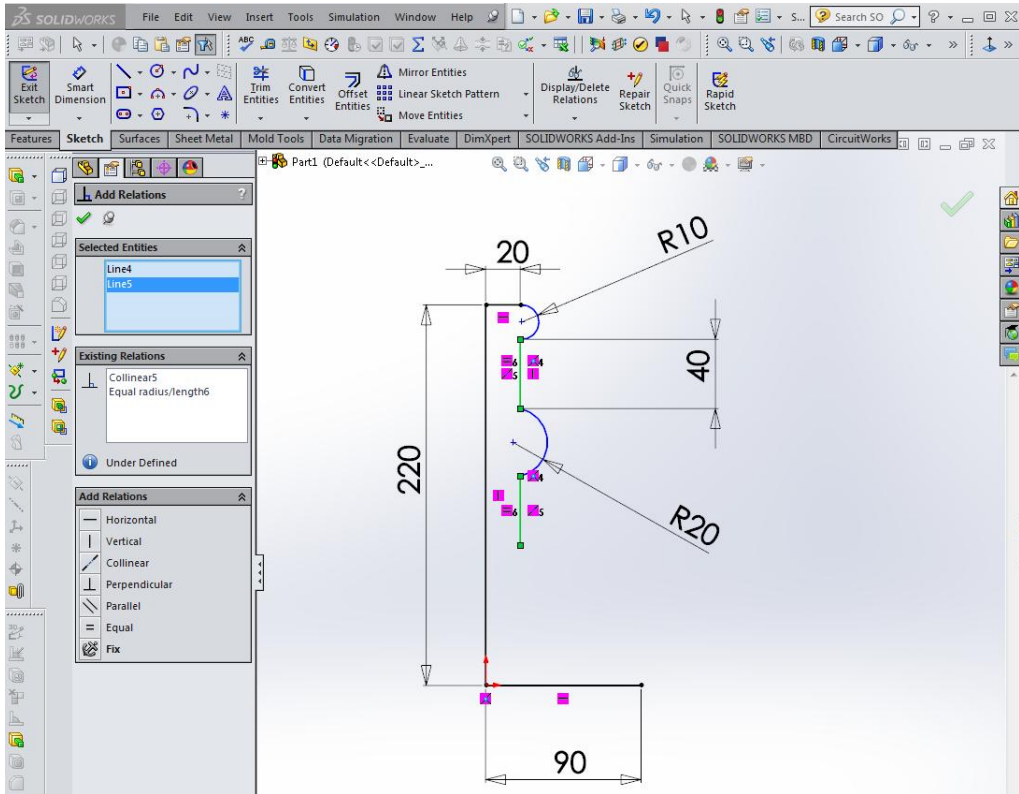


Sketching a Revolve Profile

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.
5. Click **Smart Dimension** on the Sketch toolbar.
Dimension the sketch as shown.

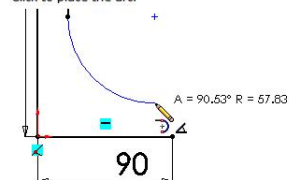
Draw Arcs, Dimension & Trim



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.



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](#)
6. Click **View > Sketch Relations** to hide the sketch relations in the graphics area.

Fully Dimensioned Profile

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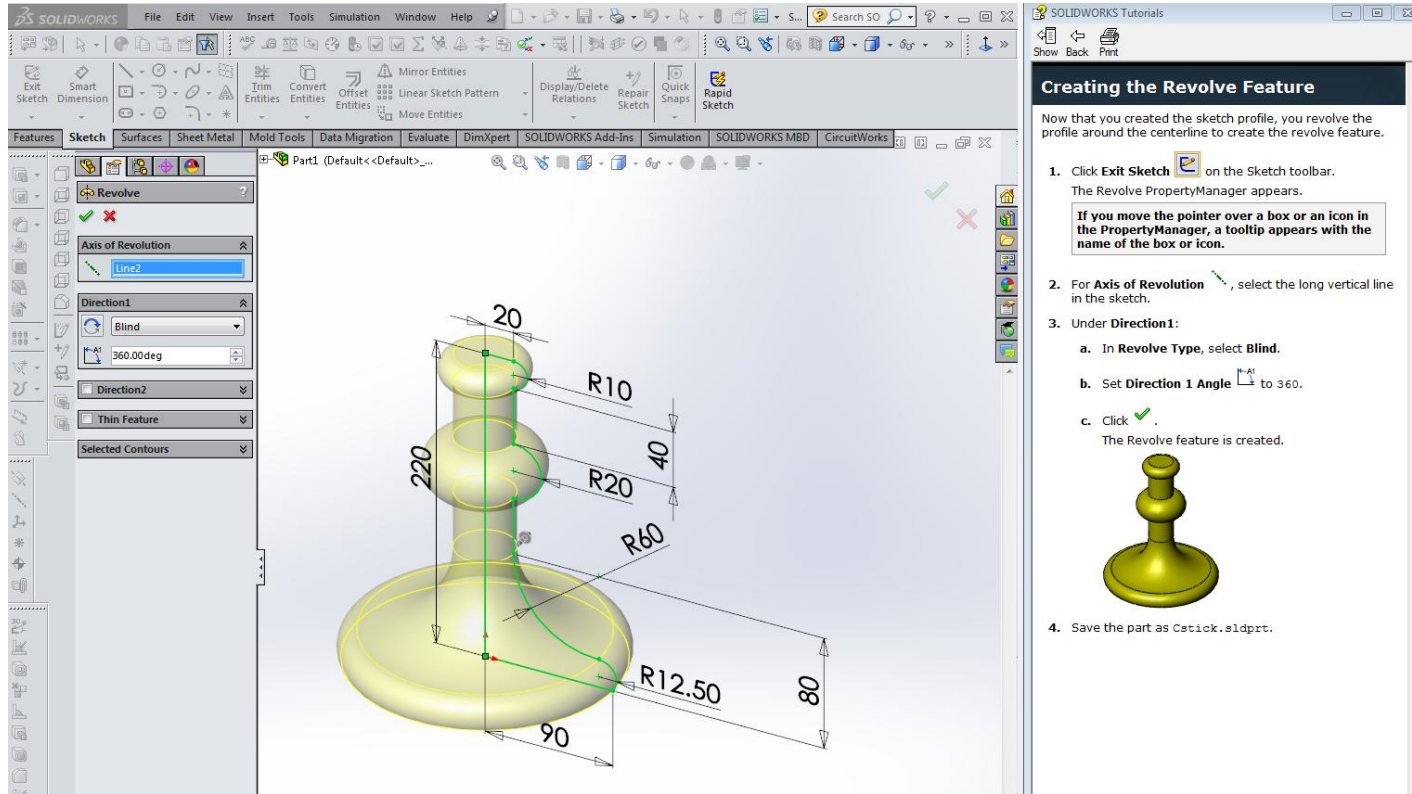
5. Sketch another tangent arc. Move the pointer until the endpoint of the arc is coincident with the endpoint of the bottom horizontal line as shown.

Video: Sketching the Tangent Arc

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

7. Dimension the rest of the sketch as shown.

Creating the Revolve Feature

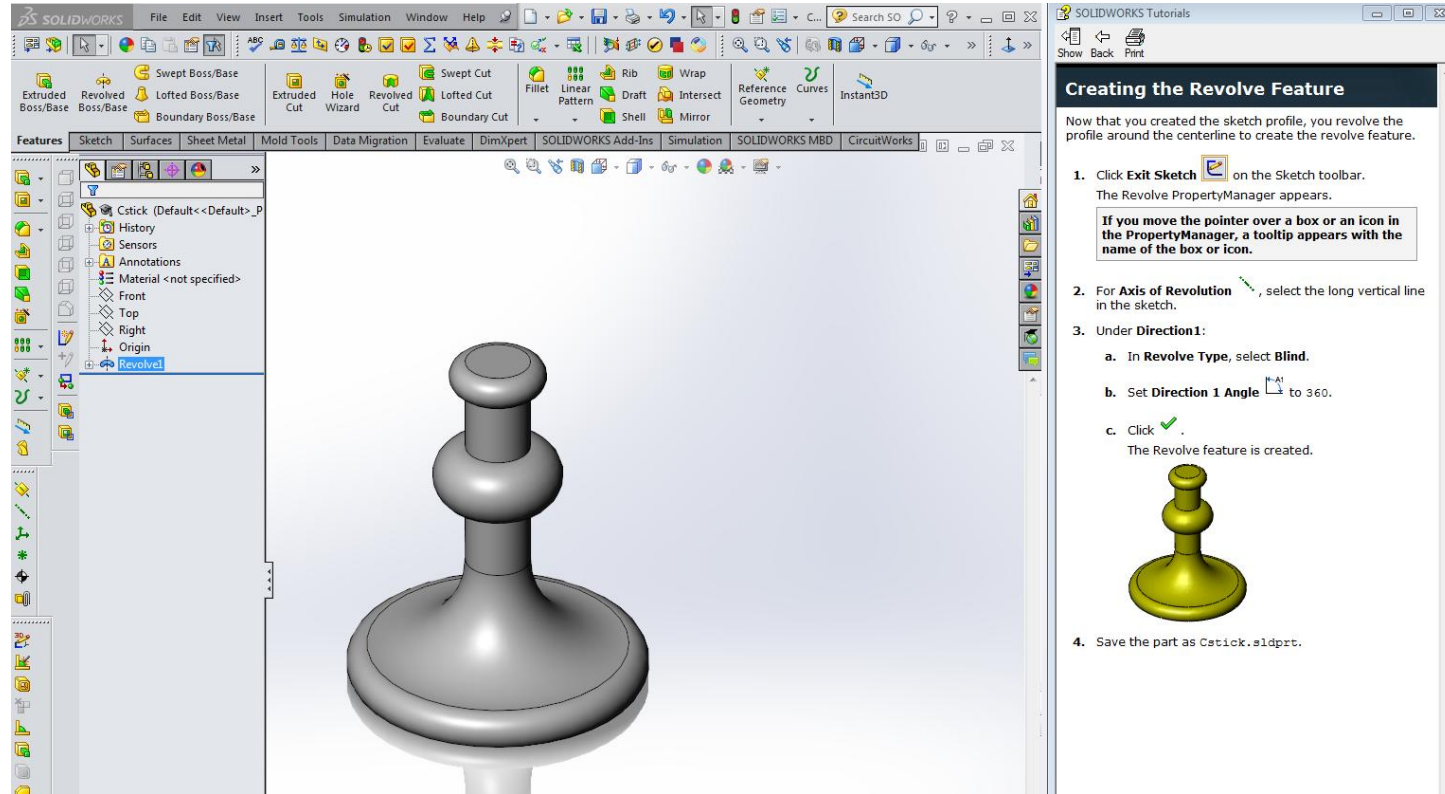


Creating the Revolve Feature

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 **OK**.
The Revolve feature is created.
4. Save the part as **Cstick.sldprt**.

Revolve Appearance








The image shows a SolidWorks software interface with a 3D model of a revolved part. The part is a grey, cylindrical object with a flared base and a smaller cylindrical section on top. The software interface includes a menu bar, a toolbar, and a feature tree on the left. The feature tree shows a 'Revolve' feature selected. The main window displays the 3D model of the part.

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Show Back Part

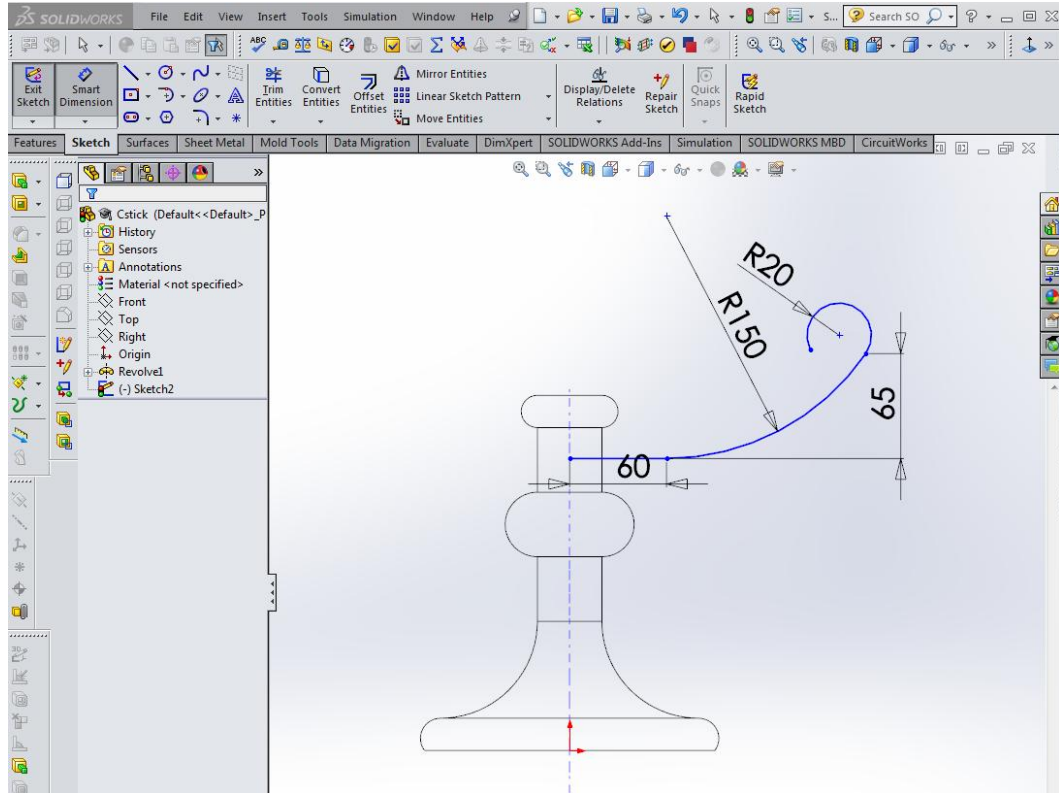
Creating the Revolve Feature

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

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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.

Completing the Sweep Path

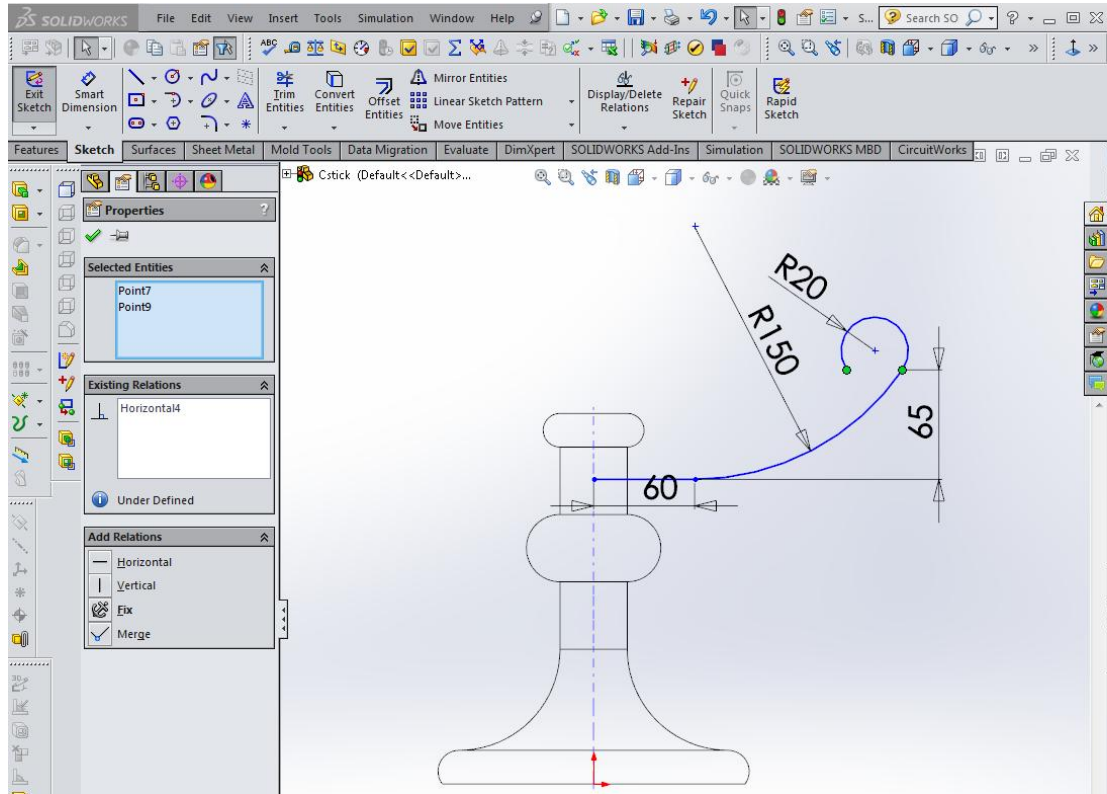


Completing the Sweep Path

1. Right-click in the graphics area and select **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 Foreshortened, 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.

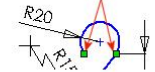
5. Right-click and select **Tangent Arc**, then sketch another arc as shown.
6. Dimension it to a radius of 20.

Add Horizontal Relationship to Arc End Points

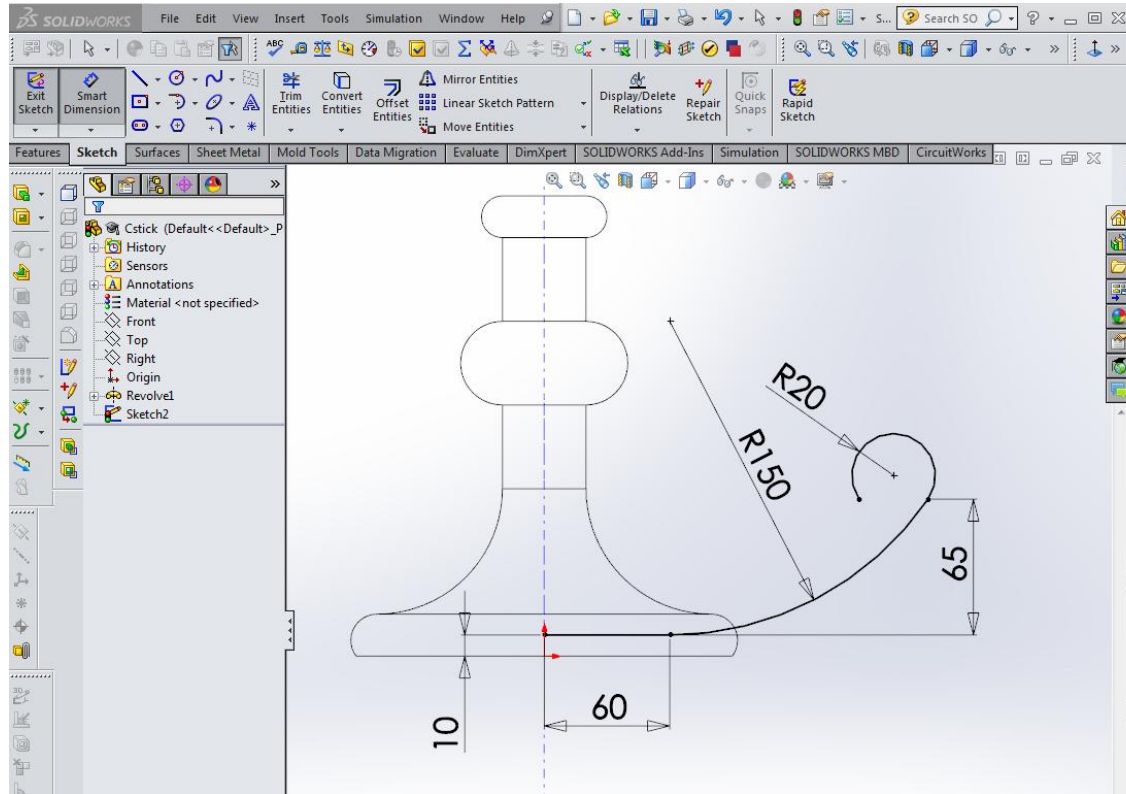


Adding Relations to the Path

Now add relations to control the sweep path.

1. Click **Select** 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** 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 Sweep Path Relative to the Base




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Dimensioning the Sweep Path

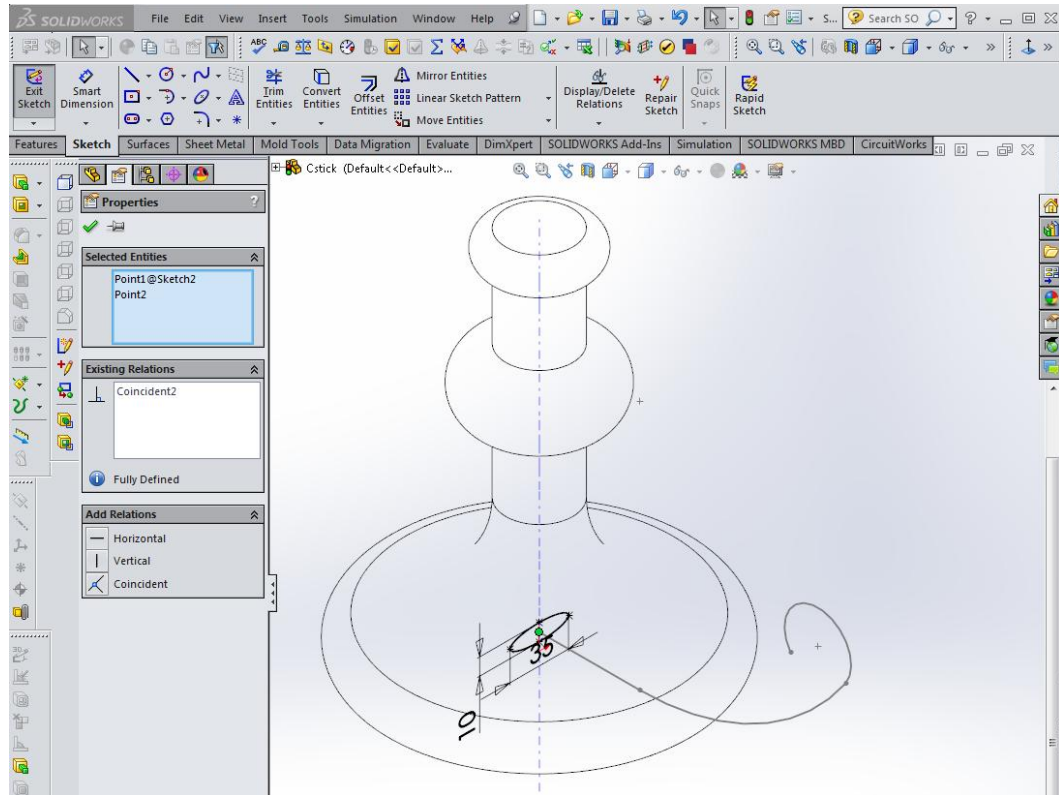
Next, dimension the sweep path with respect to the revolved base.

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.

Adding Coincident Relationship

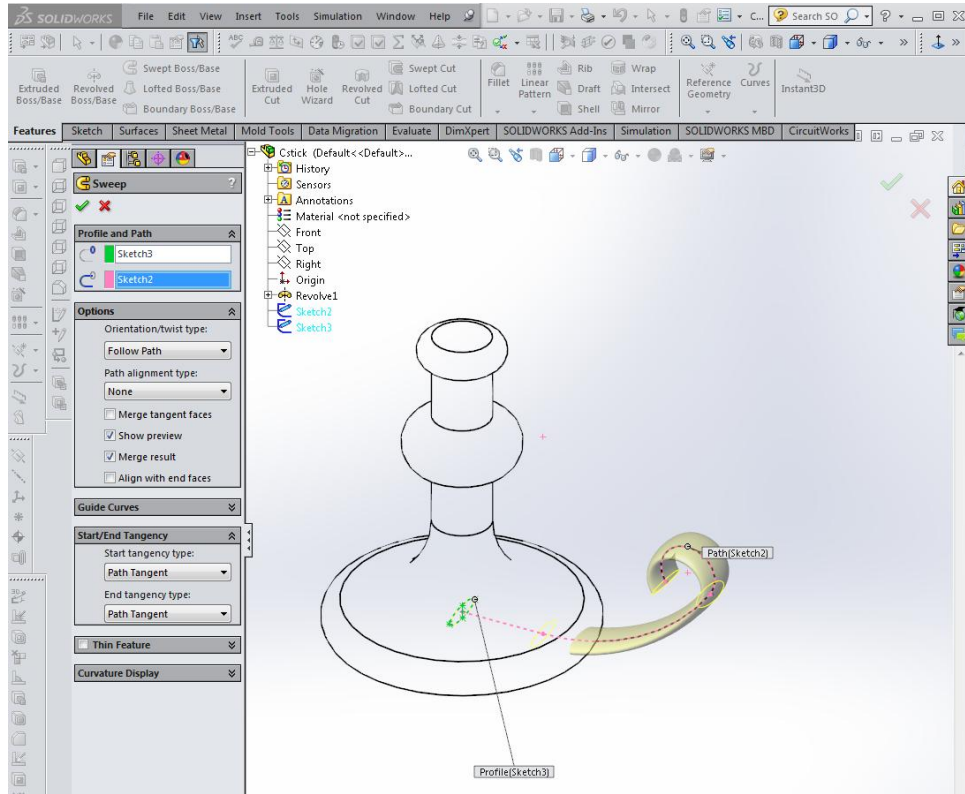


Completing the Sweep Section Sketch

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 > Temporary Axes** to hide the temporary axis.
5. Click **Exit Sketch** on the Sketch toolbar.

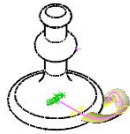
The diagrams illustrate the steps for adding a coincident relationship. The first diagram shows the selection of the center point of the ellipse and the endpoint of the horizontal line. The second diagram shows the resulting sketch with the coincident relationship applied.


Creating the Handle Sweep



Creating the Sweep

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

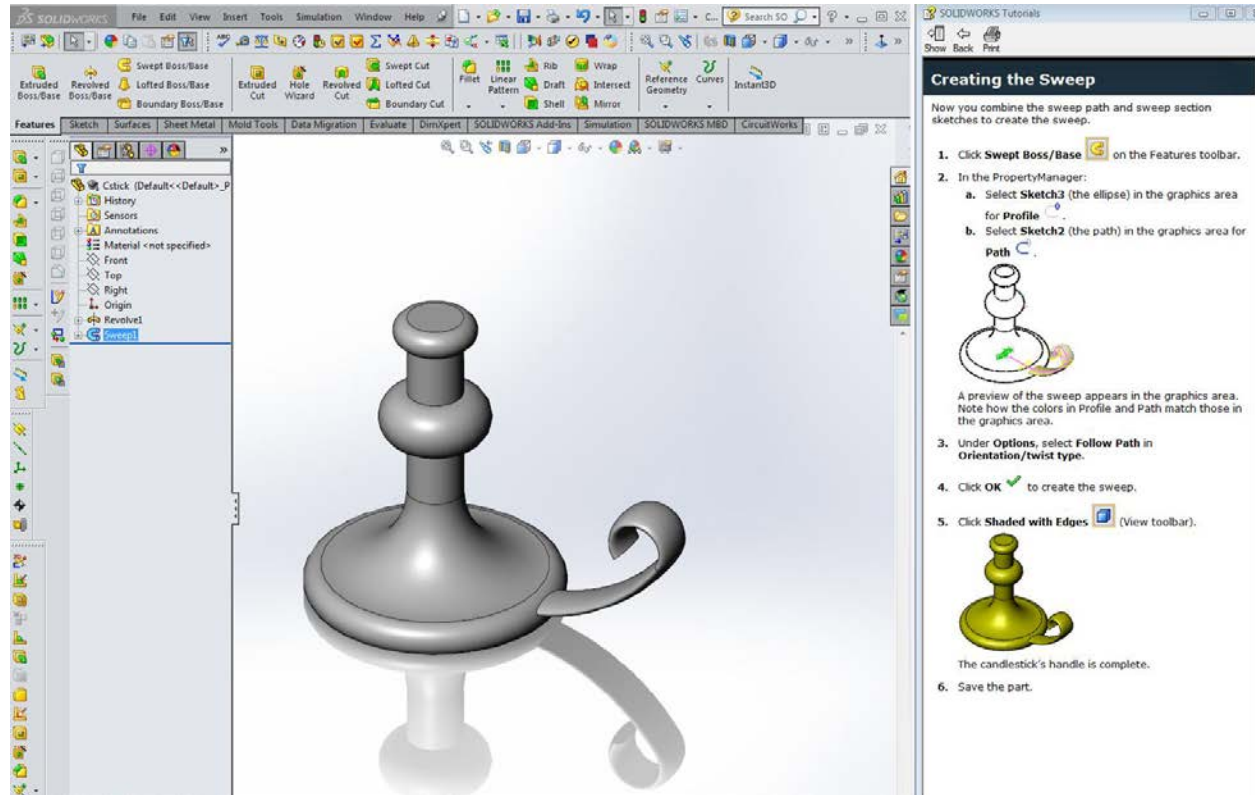
1. Click **Sweep Boss/Base** on the Features toolbar.
2. In the PropertyManager:
 - a. Select **Sketch3** (the ellipse) in the graphics area for **Profile**.
 - b. Select **Sketch2** (the path) in the graphics area for **Path**.

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 **Orientation/twist type**.
4. Click **OK** to create the sweep.
5. Click **Shaded with Edges** (View toolbar).


The candlestick's handle is complete.

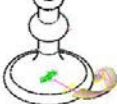
6. Save the part.

Completion of Handle




Creating the Sweep

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

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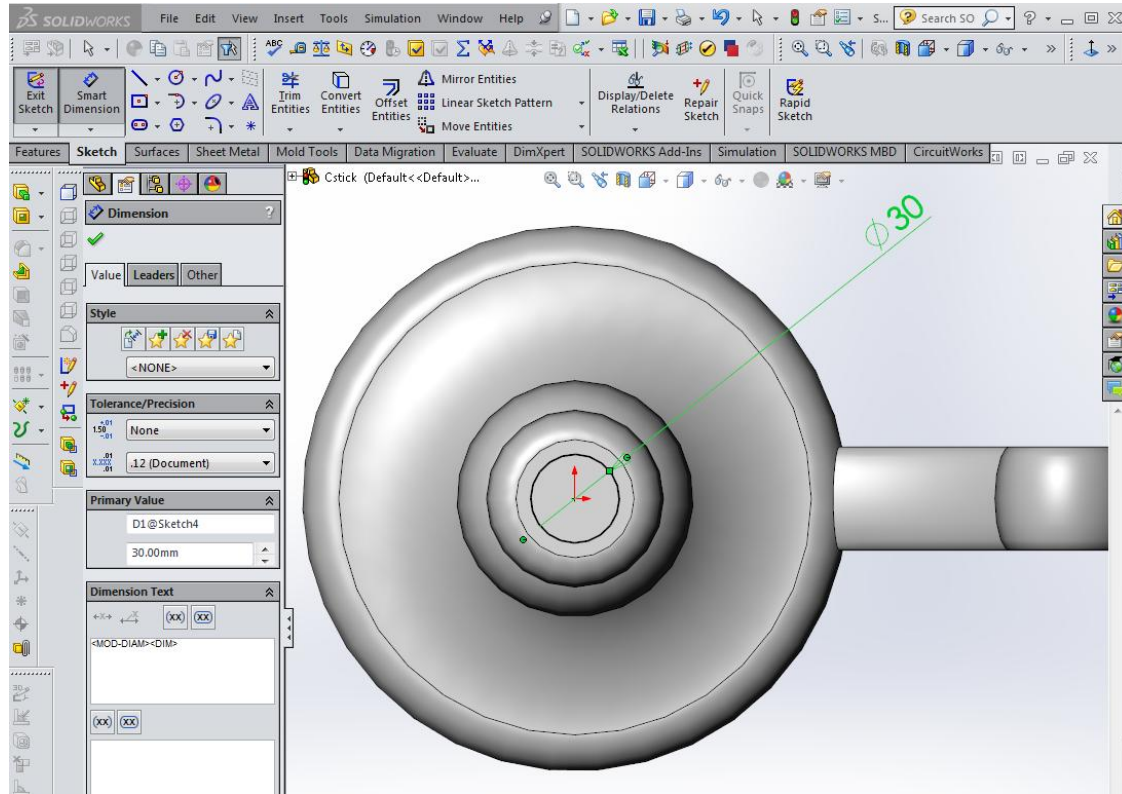
A preview of the sweep appears in the graphics area. Note how the colors in Profile and Path match those in the graphics area.
3. Under **Options**, select **Follow Path** in **Orientation/Twist** type.
4. Click **OK** to create the sweep.
5. Click **Shaded with Edges** (View toolbar).



The candlestick's handle is complete.

6. Save the part.

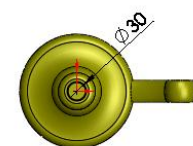
Sketch Circle for Extruded Cut Feature



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.

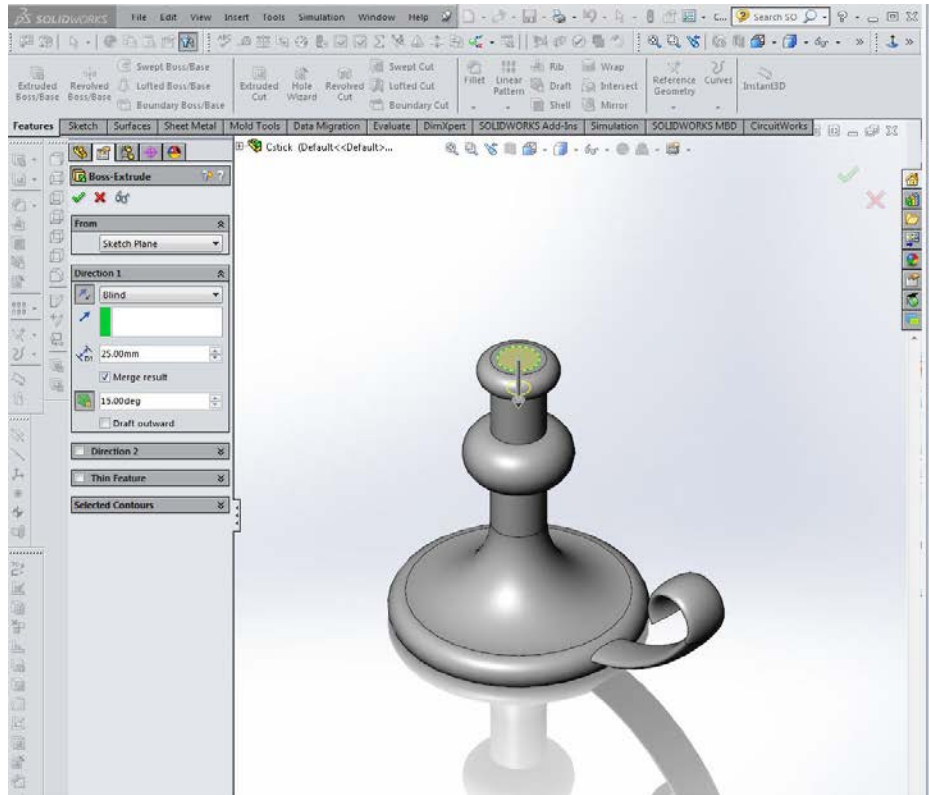


4. Click **Exit Sketch** on the Sketch toolbar. The Cut-Extrude PropertyManager opens.
5. Click **Isometric** (Standard Views toolbar).
6. In the PropertyManager, under **Direction 1**:
 - a. Select **Blind in End Condition**.

Click **Reverse Direction** if necessary to make the arrow point down.





Extruded Cut with Draft Angle

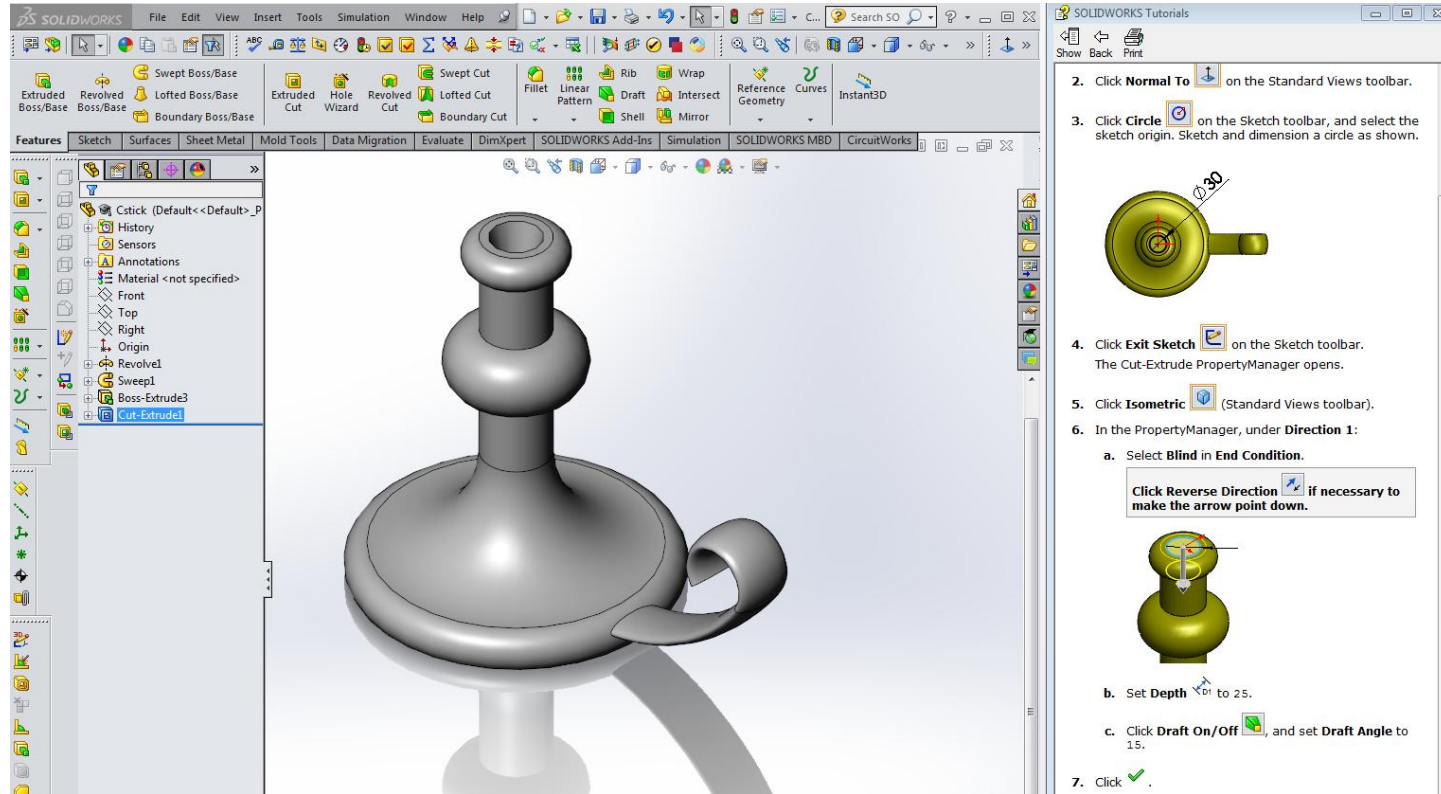


Completing the Part



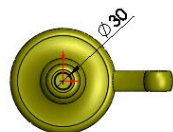


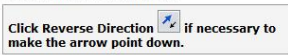



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2. Click **Normal To** on the Standard Views toolbar.
3. Click **Circle** on the Sketch toolbar, and select the sketch origin. Sketch and dimension a circle as shown.

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.

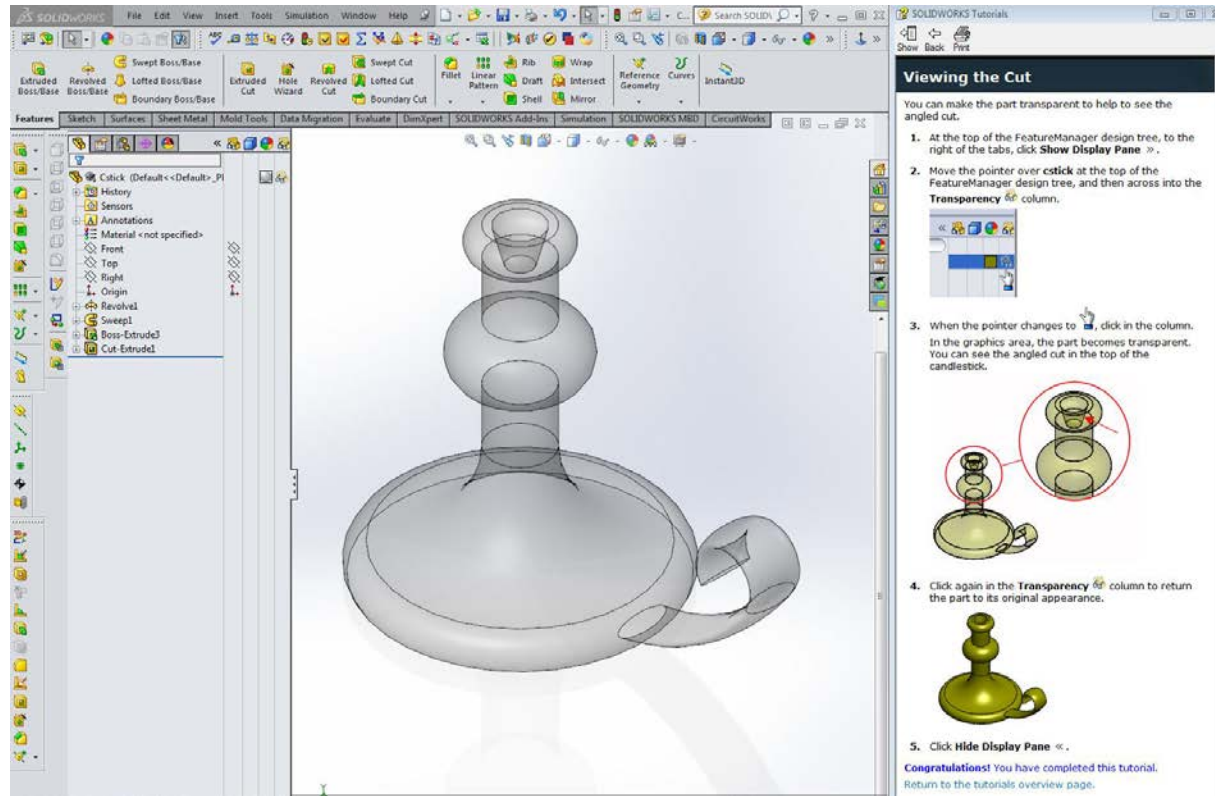
Cut Added for Candle



The screenshot displays the SolidWorks CAD environment. The main window shows a 3D model of a candle with a cut feature applied to its side. The left-hand side contains the Feature Tree, where the 'Cut-Extrude1' feature is highlighted. The top ribbon shows various modeling tools. On the right, a tutorial panel titled 'SOLIDWORKS Tutorials' provides step-by-step instructions for creating the cut.


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**.

 - b. Set **Depth**  to 25.
 - c. Click **Draft On/Off** , and set **Draft Angle** to 15.
7. Click .

Making Cut Transparent



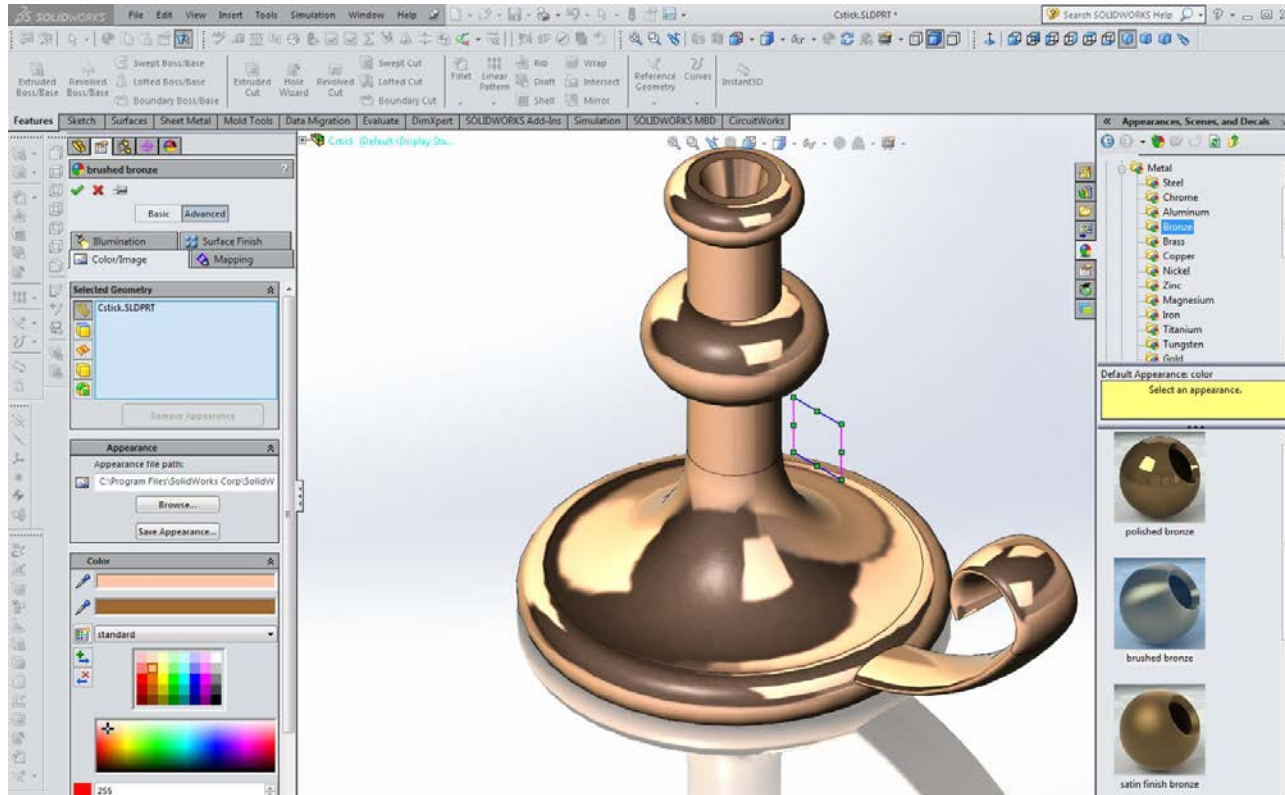
Viewing the Cut

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 **cutick** 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**.

Congratulations! You have completed this tutorial.
Return to the tutorials overview page.

Adding a Satin Bronze Finish



Summary

- ▶ Topics Covered in this exercise:
 - Sketching a revolve profile with lines, arcs, dimensioning, and trimming.
 - Adding dimensional relationships.
 - Creating a revolve.
 - Creating a sweep path and coincidental relationship.
 - Sweeping along the path.
 - Making an extruded-cut with draft.
 - Making a transparent view to see the cuts.
 - Making a Real View appearance.