SolidWorks Part 2

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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

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
Fully Dimensioned Profile
Creating 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 Direction 1:
   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 Castile.xipt.
Revolve Appearance

Creating the Revolve Feature

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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 Castile_adaptor.
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 not out of view, click the Leaders tab in the Dimension PropertyManager.
   - Click **Foreshortened** and 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

1. Click Select All on the Standard toolbar, then hold down Ctrl and select the endpoints of the tangent arc you just sketched.

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 √.

The Properties PropertyManager appears. The two endpoints are listed under Selected Entities.
Dimensioning Sweep Path Relative to the Base

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

2. Click Exit Sketch on the Sketch toolbar.

The sweep path is fully defined.
Adding Coincident Relationship

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.
Creating the Handle Sweep

1. Click "Sweep Boss/Base" on the Features toolbar.
2. In the PropertyManager:
   a. Select Sketch1 (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 skicthe 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 Sketch 2 (the path) in the graphics area for Path.
3. Under Options, select Follow Path in Orientation/Intersect type.
4. Click OK to create the sweep.
5. Click "Shaded with Edges" (View toolbar).
6. The candlestick's handle is complete.
7. Save the part.
Sketch Circle for Extruded Cut Feature

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Extruded Cut with Draft Angle
Cut Added for Candle
Making Cut Transparent

Viewing the Cut

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

2. Move the pointer over each of 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 candlewick.

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.

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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.