Lesson 9: Revolve and Sweep Features

Goals of This Lesson

Upon successful completion of this lesson, your students will be able to create and modify the following parts and assembly:

Resources for This Lesson

This lesson plan corresponds to the Revolves and Sweeps module in the SolidWorks Online Tutorials. For more information about the Online Tutorials, See “Online Tutorials” on page v.
Lesson 9: Revolve and Sweep Features

Active Learning Exercises — Creating a Candlestick

Create the candlestick. Follow the instructions in the Building Models: Revolves and Sweeps module in the SolidWorks Online Tutorials.

The part name is Cstick.sldprt. However, throughout this lesson, we will refer to it as “candlestick” because that makes more sense.

5 Minute Assessment

1. What features did you use to create the candlestick?

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2. What special piece of sketch geometry is useful, but not required for a revolve feature?

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3. Unlike an extruded feature, a swept feature requires a minimum of two sketches. What are these two sketches?

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4. What information does the pointer provide while sketching an arc?

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5. Examine the three illustrations at the right. Which one is not a valid sketch for a revolve feature?

Why?

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Lesson 9: Revolve and Sweep Features

Exercises and Projects — Creating a Candle to Fit the Candlestick

Task 1— Revolve Feature

Design a candle to fit the candlestick.

☐ Use a revolve feature as the base feature.

☐ Taper the bottom of the candle to fit into the candlestick.

☐ Use a sweep feature for the wick.

Question:

What other features could you use to create the candle? Use a sketch to illustrate your answer if necessary.

Answer:

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Question:

Would there be any benefit to using a design table to create the candle?

Answer:

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Task 2 — Create an Assembly

Create a candlestick assembly.

Task 3 — Create a Design Table

You work for a candle manufacturer. Use a design table to create 380 mm, 350 mm, 300 mm, and 250 mm candles.

<table>
<thead>
<tr>
<th>A</th>
<th>B</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Design Table for: candle</td>
</tr>
<tr>
<td>2</td>
<td>Length@Sketch1</td>
</tr>
<tr>
<td>3</td>
<td>380 mm candle</td>
</tr>
<tr>
<td>4</td>
<td>350 mm candle</td>
</tr>
<tr>
<td>5</td>
<td>300 mm candle</td>
</tr>
<tr>
<td>6</td>
<td>250 mm candle</td>
</tr>
</tbody>
</table>
More to Explore — Design and Model a Mug

Design and model a mug. This is a rather open-ended assignment. You have an opportunity to express your creativity and ingenuity. The design of a mug can vary from the simple to the complex. A couple of examples are shown at the right.

There are two specific requirements:

- Use a revolve feature for the body of the mug.
- Use a swept feature for the handle.

Task 4

How much coffee does the mug shown at the right hold?

Given:

- Inside Diameter = 2.50”
- Overall height of the mug = 3.75”
- Thickness of the bottom = 0.25”
- Coffee cups are not filled to the brim. Allow 0.5” space at the top.

Answer:
Conversion:

A cup of coffee in the US is sold by the fluid ounce, not by the cubic inch. How many ounces does the mug hold?

Given:

1 gallon = 231 in³
128 ounces = 1 gallon

Answer:

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_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________
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Modify the outlet plate that you created earlier in Lesson 2.

- Edit the sketch for the circular cuts that form the openings for the outlet. Create new cuts using the sketch tools. Apply what you have learned about Link Values and geometric relations to properly dimension and constrain the sketch.

- Add a swept boss feature to the back edge.
  - The sweep section is a 90° arc.
  - The radius of the arc is equal to the length of the model edge as shown in the accompanying illustration.
  - Use geometric relations to fully define the sweep section sketch.
  - The sweep path is made up of the four rear edges of the part.
  - Use Convert Entities to create the sweep path.

- The desired result is shown in the illustration at the right.
More to Explore — Use Revolve Feature to Design a Top

Use a revolve feature to create a toy top of your own design.
Lesson Summary

- A Revolve feature is created by rotating a 2D profile sketch around an axis of revolution.
- The profile sketch can use a sketch line (that is part of the profile) or a centerline as the axis of revolution.
- The profile sketch cannot cross the axis of revolution.

![Good](image1.png) ![Good](image2.png) ![No Good](image3.png)

- The Sweep feature is created by moving a 2D profile along a path.
- The Sweep feature requires two sketches:
  - Sweep Path
  - Sweep Section
- Draft tapers the shape. Draft is important in molded, cast, or forged parts.
- Fillets are used to smooth edges.