SolidWorks
workshop

■ Workbook (Bring to each workshop)

Integrated Teaching and Learning Laboratory & Program
College of Engineering and Applied Science
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Introduction

Online Tutorials

The *SolidWorks Teacher Guide and Student Courseware* is a companion resource and supplement for the SolidWorks Online Tutorials. Many of the exercises in the *SolidWorks Student Workbook* use material from the Online Tutorials.

Accessing the Tutorials

To start the Online Tutorials, click **Help, Online Tutorial**. The SolidWorks window is resized and a second window will appears next to it with a list of the available tutorials. As you move the pointer over the links, an illustration of the tutorial will appear at the bottom of the window. Click the desired link to start that tutorial.
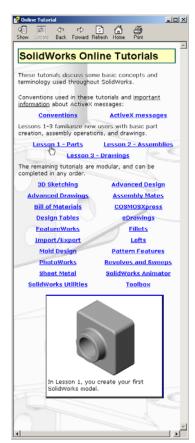
Conventions

Set your screen resolution to 1280x1024 for optimal viewing of the tutorials.

The following icons appear in the tutorials:

Next ≫ Moves to the next screen in the tutorial.

☀ Represents a note or tip. It is not a link; the information is below the icon. Notes and tips provide time-saving steps and helpful hints.

🖉 You can click most toolbar buttons that appear in the lessons to flash the corresponding SolidWorks button.The first time you click the button, an ActiveX control message appears: `An ActiveX control on this page might be unsafe to interact with other parts of the page. Do you want to allow this interaction?` This is a standard precautionary measure. The ActiveX controls in the Online Tutorials will <u>not</u> harm your system. If you click **No**, the scripts are disabled for that topic. Click **Yes** to run the scripts and flash the button.

➲ **Open File** or **Set this option** automatically opens the file or sets the option.

🔍 **A closer look at...** links to more information about a topic. Although not required to complete the tutorial, it offers more detail on the subject.

❓ **Why did I...** links to more information about a procedure, and the reasons for the method given. This information is not required to complete the tutorial.

Printing the Tutorials

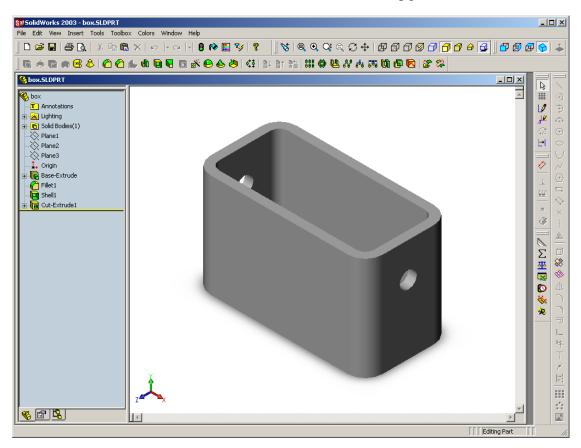If you like, you can print the Online Tutorials by following this procedure:

1 On the tutorial navigation toolbar, click **Show**.

This displays the table of contents for the Online Tutorials.

2 Right-click the book representing the lesson you wish to print and select **Print** from the shortcut menu.

The **Print Topics** dialog box appears.

3 Select **Print the selected heading and all subtopics**, and click **OK**.

4 Repeat this process for each lesson that you want to print.

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Lesson 2: Basic Functionality

Goals of This Lesson

❑ Upon successful completion of this lesson, you will understand the basic functionality of SolidWorks software and be able to create the following part:

Active Learning Exercise — Creating a Basic Part

Use SolidWorks to create the box shown at the right.
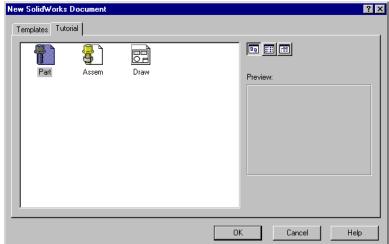
The step-by-step instructions are given below.

Create a New Part Document

1 Create a new part. Click ▢ on the Standard toolbar.

The **New SolidWorks Document** dialog box appears.

2 Click the **Tutorial** tab.

3 Select the **Part** icon.

4 Click **OK**.

A new part document window appears.

Base Feature

The Base feature requires:

❑ Sketch plane – `Front` (default plane)

❑ Sketch profile – 2D Rectangle

❑ Feature type – Extruded boss feature

Open a Sketch

5 Open a 2D sketch. Click 🖊 on the Sketch toolbar.

The sketch opens on the `Front` plane. `Front` is the default plane listed in the FeatureManager design tree.

Confirmation Corner

When many SolidWorks commands are active, a symbol or a set of symbols appears in the upper right corner of the graphics area. This area is called the **Confirmation Corner**.

Sketch Indicator

When a sketch is active, or open, a symbol appears in the confirmation corner that looks like the **Sketch** tool. It provides a visual reminder that you are active in a sketch. Clicking the symbol exits the sketch saving your changes. Clicking the red X exits the sketch discarding your changes.
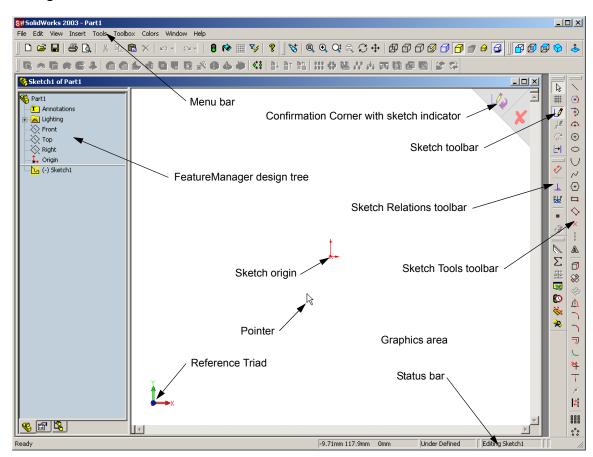
When other commands are active, the confirmation corner displays two symbols: a check mark and an X. The check mark executes the current command. The X cancels the command.

Overview of the SolidWorks Window

❑ A sketch origin appears in the center of the graphics area.

❑ The Sketch Tools and Sketch Relations toolbars are displayed.

❑ "Editing Sketch" appears in the status bar at the bottom of the screen.

❑ `Sketch1` appears in the FeatureManager design tree.

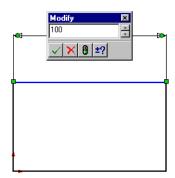❑ The status bar shows the position of the pointer, or sketch tool, in relation to the sketch origin.

Sketch a Rectangle

6 Click ☐ on the Sketch Tools toolbar.

7 Click the sketch origin to start the rectangle.

8 Move the pointer up and to the right, to create a rectangle.

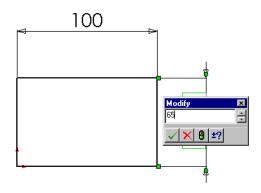**9** Click the mouse button again to complete the rectangle.

Add Dimensions

1 Click **Dimension** ⬲ on the Sketch Relations toolbar.
The pointer shape changes to ⬲.

2 Click the top line of the rectangle.

3 Click the dimension text location above the top line.
The **Modify** dialog box is displayed.

4 Enter **100**. Click ✔ or press **Enter**.

5 Click the right edge of the rectangle.

6 Click the dimension text location. Enter **65**.
Click ✔.
The top segment and the remaining vertices are displayed in black. The status bar in the lower-right corner of the window indicates that the sketch is fully defined.

Changing the Dimension Values

The new dimensions for the box are 100mm x 60mm. Change the dimensions. Use the **Select** tool.

7 Click **Select** ⬲ on the Sketch toolbar.

8 Double-click **65**.
The **Modify** dialog box appears.

9 Enter **60** in the **Modify** dialog box.

10 Click ✔.

Extrude the Base Feature.

The first feature in any part is called the *Base Feature*. In this exercise, the base feature is created by extruding the sketched rectangle.

11 Click **Extruded Boss/Base** ⬲ on the Features toolbar.
The **Extrude Feature** PropertyManager appears. The view of the sketch changes to isometric.

12 Preview graphics.

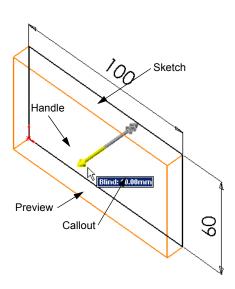A preview of the feature is shown at the default depth.

Handles ✏️ appear that can be used to drag the preview to the desired depth. The handles are colored yellow for the active direction and gray for inactive direction. A callout shows the current depth value.

Click on the screen to set the preview into **Shaded**

mode. The cursor changes to 🖱️. If you want to create the feature now, click the right mouse button. Otherwise, you can make additional changes to the settings. For example, the depth of extrusion can be changed by dragging the dynamic handle with the mouse or by setting a value in the PropertyManager.

13 Extrude feature settings.

Change the settings as shown.

- End Condition = **Blind**

- $\cdot$ 📐 (Depth) = **50**

14 Create the extrusion. Click **OK** ✅.

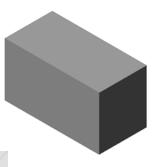The new feature, Extrude1, is displayed in the FeatureManager design tree.
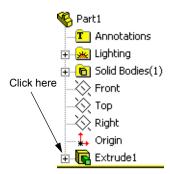
TIP:

The **OK** button ✅ on the PropertyManager is just one way to complete the command.

A second method is the set of **OK/Cancel** buttons in the confirmation corner of the graphics area.

A third method is the right-mouse shortcut menu that includes **OK**, among other options.

15 Click the plus sign ⊞ beside Extrude1 in the
FeatureManager design tree. Notice that Sketch1 — which
you used to extrude the feature — is now listed under the
feature.

View Display

Change the display mode. Click **Hidden Lines Visible**
on the View toolbar.

Hidden Lines Visible allows you to select hidden back
edges of the box.

Save the Part

1 Click **Save** on the Standard toolbar, or click **File,
Save.**

The **Save As** dialog box appears.

2 Type box for the filename. Click **Save**.

The .sldprt extension is added to the filename.

The file is saved to the current directory. You can use the Windows browse button to
change to a different directory.

Round the Corners of the Part

Round the four corner edges of the box. All rounds have the same
radius (10mm). Create them as a single feature.

3 Click **Fillet** on the Features toolbar.
The **Fillet** PropertyManager appears.

4 Enter **10** for the **Radius.**
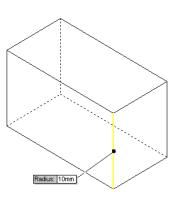Leave the remaining settings at their default values.

5 Click the first corner edge.

The faces, edges, and vertices are highlighted as you move the pointer over them.
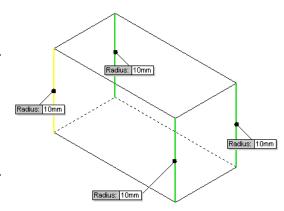
When you select the edge, a callout `Radius: 10mm` appears.

6 Identify selectable objects. Notice how the pointer changes shapes:

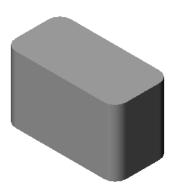Edge: ⊹ ‖   Face: ⊹ ▱ Vertex: ⊹ ▫

7 Click the second, third and fourth corner edges.

> **Note:** Normally, a callout only appears on the *first* edge you select. This illustration has been modified to show callouts on each of the four selected edges. This was done simply to better illustrate which edges you are supposed to select.

8 Click ✅.

`Fillet1` appears in the FeatureManager design tree.
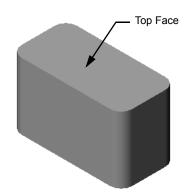
Hollow Out the Part

Remove the top face using the Shell feature.

9 Click ⬛ on the Features toolbar.

The **Shell Feature** PropertyManager appears.

10 Enter **5** for **Thickness**.

11 Click the top face.

Top Face

12 Click **OK**
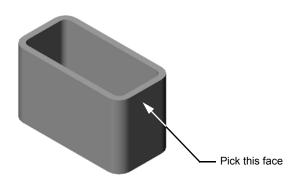
Wall Thickness

Extruded Cut Feature

The Extruded Cut feature removes material. To make an extruded cut requires a:

❏ Sketch plane – In this exercise, the face on the right-hand side of the part.
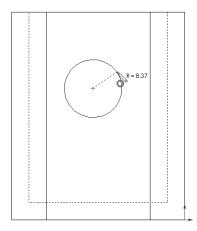
❏ Sketch profile – 2D circle

Open a Sketch

13 To select the sketch plane, click the right-hand face of the box.

14 Click 🔷 on the Standard Views toolbar. The view of the box turns. The selected model face is facing you.

15 Open a 2D sketch. Click 📝 on the Sketch toolbar.

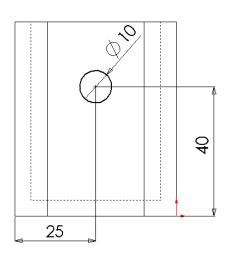Pick this face

Sketch the Circle

1 Click ⊙ on the Sketch Tools toolbar.

2 Position the pointer where you want the center of the circle. Click the left mouse button.

3 Drag the pointer to sketch a circle.

4 Click the left mouse button again to complete the circle.

Dimension the Circle

Dimension the circle to determine its size and location.

5 Click 🖉 on the Sketch Relations toolbar.

6 Dimension the diameter. Click on the circumference of the circle. Click a location for the dimension text in the upper right corner. Enter **10**.

7 Create a horizontal dimension. Click the circumference of the circle. Click the left most vertical edge. Click a location for the dimension text below the bottom horizontal line. Enter **25**.

8 Create a vertical dimension. Click the circumference of the circle. Click the bottom most horizontal edge. Click a location for the dimension text to the right of the sketch. Enter **40**.

Extrude the Sketch

1 Click 🖾 on the Features toolbar.
The **Extrude Cut Feature** PropertyManager appears.

2 Select **Through All** for the end condition.

3 Click ✔.

4 Results.

The cut feature is displayed.

Rotate the View

Rotate the view in the graphics area to display the model from different angles.

5 Rotate the part in the graphics area. Press and hold the middle mouse button. Drag the pointer up/down or left/right. The view rotates dynamically.

6 Display the Isometric view. Click 🔷 on the Standard Views toolbar.

Save the Part

1 Click 💾 on the Standard toolbar

2 Click **File, Exit** on the Main menu.

Exercises and Projects — Designing a Switch Plate

Switch plates are required for safety. They cover live electrical wires and protect people from electric shock. Switch plates are found in every home and school. They incorporate simple and complex designs.
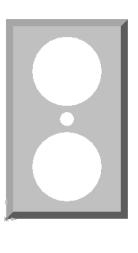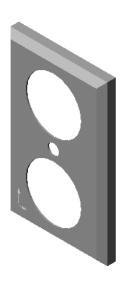
⚠ Caution: Do not use metal rulers near switch plates attached to a live wall outlet.

Tasks

1 Measure a single light plate switch cover.

Answer: _____

_____

_____

2 Using paper and pencil, manually sketch the light plate switch cover.

3 Label the dimensions.

4 What is the base feature for the light plate switch cover?

Answer: _____

_____

_____

_____

5 Create a simple single light switch cover using SolidWorks. The filename for the part is `switchplate`.

6 What features are used to develop the `switchplate`?

Answer: _____

_____

_____

_____

_____

7 Create a simplified duplex outlet cover plate. The filename for the part is `outletplate.`

8 Save the parts. They will be used in later lessons.

Lesson Summary

❑ SolidWorks is design automation software.
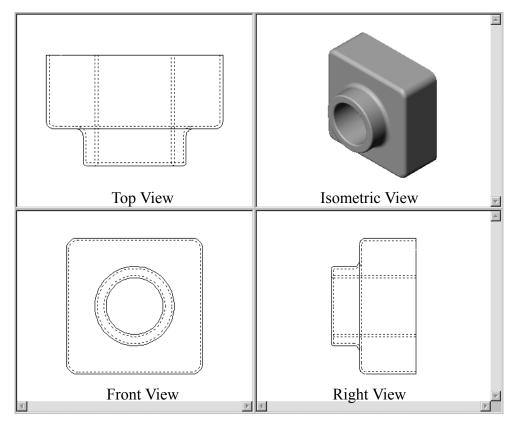
❑ The SolidWorks model is made up of:
Parts
Assemblies
Drawings

❑ Features are the building blocks of a part.

<div align="right">

3

</div>

Lesson 3: The 40-Minute Running Start

Goals of This Lesson

❑ You will be able to create and modify the following part:

Top View	Isometric View

Front View	Right View

Before Beginning This Lesson

❑ Complete the previous lesson — Basic Functionality.

Resources for This Lesson

This lesson plan corresponds to *Lesson 1 – Parts* in the SolidWorks Online Tutorials. For more information about the Online Tutorials, See "Online Tutorials" on page v.

Active Learning Exercise — Create a Part

Follow the instructions in *Lesson 1 – Parts* of the SolidWorks
Online Tutorial. In this lesson you will create the part shown at the
right. The part name is `Tutor1.sldprt.`

5 Minute Assessment

1 What features did you use to create `Tutor1`? _____

   _____

2 What does the Fillet feature do?_____

   _____

3 What does the Shell feature do?_____

   _____

4 Name three view commands in SolidWorks. _____

   _____

5 Where are the display buttons located? _____

   _____

6 Name the three SolidWorks default planes. _____

   _____

7 The SolidWorks default planes correspond to what principle drawing views? _____

   _____

   _____

   _____

   _____

8 True or False. In a fully defined sketch, geometry is displayed in black. _____

   _____

9 True or False. It is possible to make a feature using an over defined sketch. _____

   _____

10 Name the primary drawing views used to display a model._____
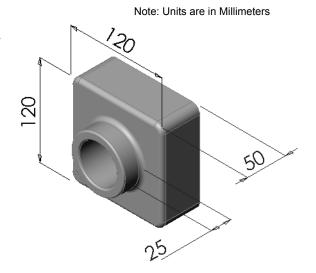
   _____

   _____

Exercises and Projects — Modifying the Part

Task 1— Converting Dimensions

The design for `Tutor1` was created in Europe. `Tutor1` will be manufactured in the US. Convert the overall dimensions of `Tutor1` from millimeters to inches.

Note: Units are in Millimeters

Given:

❑ Conversion: 25.4 mm = 1 inch

❑ Base width = 120 mm

❑ Base height = 120 mm

❑ Base depth = 50 mm

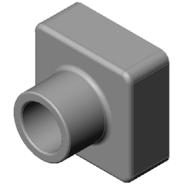❑ Boss depth = 25 mm

Answer:

_____

_____

_____

_____

_____

Task 2— Calculating the Modification

The current overall depth of `Tutor1` is 75 mm. Your customer requires a design change. The new required overall depth is 100 mm. The Base depth must remain fixed at 50 mm. Calculate the new Boss depth.

Given:

❑ New overall depth = 100 mm

❑ Base depth = 50 mm

Answer:

_____

_____

_____

_____

_____

Task 3— Modifying the Part

Using SolidWorks, modify `Tutor1` to meet the customer's requirements. Change the depth of the Boss feature such that the overall depth of the part equals 100 mm. Save the modified part under a different name.
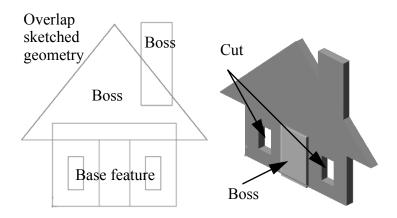
Task 4— Calculating Material Volume

Material volume is an important calculation for designing and manufacturing parts. Calculate the volume of the Base feature in mm^3 for `Tutor1`.

Answer:

_____

_____

_____

_____

_____

_____

_____

Task 5— Calculating the Volume of the Base feature

Calculate the volume of the Base feature in cm^3.

Given:

❑ 1cm = 10mm

Answer:

_____

_____

_____

_____

_____

_____

_____

_____

_____

Task 1   `house.sldprt`

Overlap
sketched
geometry

Boss

Cut

Boss

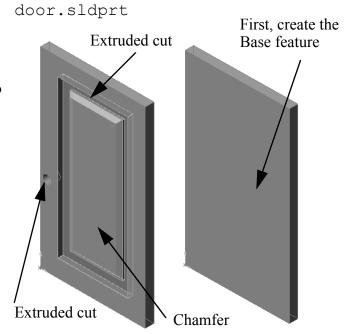Base feature

Boss

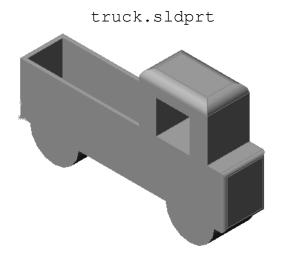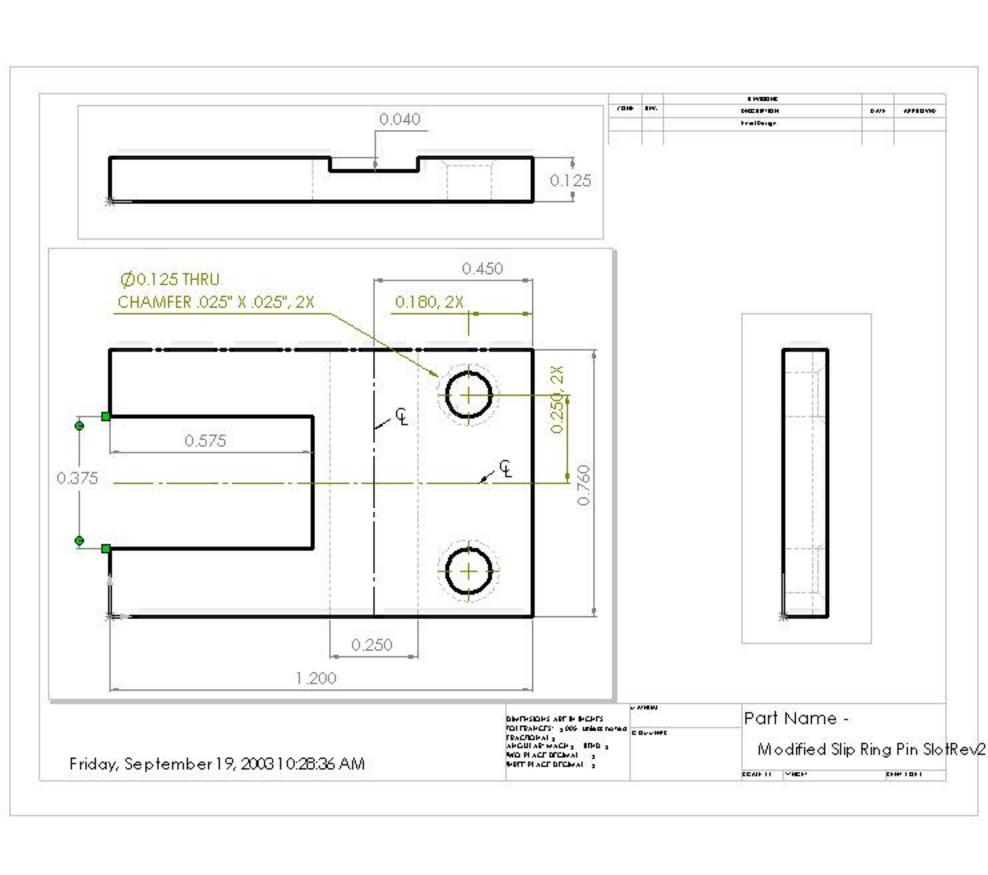**Task 2**   `door.sldprt`
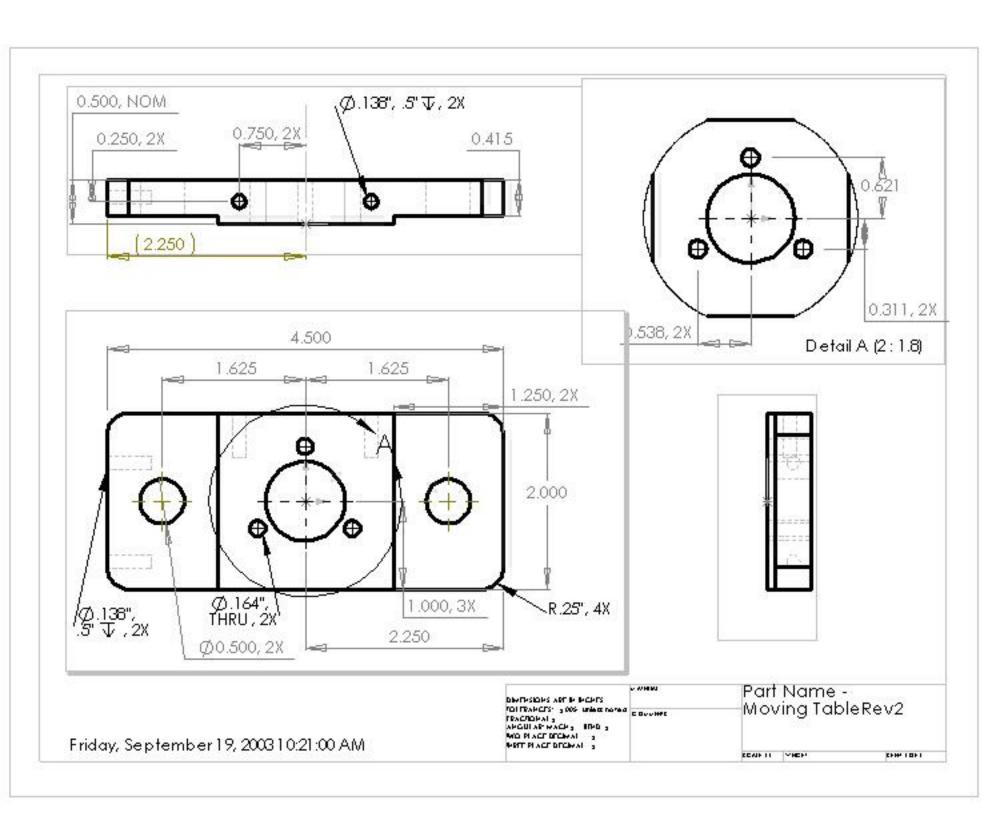
The Chamfer feature is a new
feature. The chamfer feature
removes material along an
edge. It works very similarly to
a fillet except the result is a
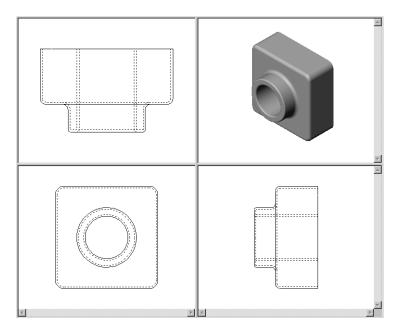beveled edge rather than a
rounded edge.

Extruded cut

First, create the
Base feature

Extruded cut

Chamfer

Task 3   `truck.sldprt`

0.040

0.125

Ø0.125 THRU
CHAMFER .025" X .025", 2X

0.450

0.180, 2X

0.250, 2X

0.760

0.575

C_L

C_L

0.375

0.250

1.200

Part Name -

Modified Slip Ring Pin SlotRev2

Friday, September 19, 2003 10:28:36 AM

0.500, NOM

0.250, 2X

0.750, 2X

Ø.138", .5" ⊽, 2X

0.415

(2.250)

4.500

1.625

1.625

1.250, 2X

2.000

Ø.138",
.5" ⊽, 2X

Ø.164",
THRU, 2X

1.000, 3X

R.25", 4X

Ø0.500, 2X

2.250

0.621

0.538, 2X

0.311, 2X

Detail A (2 : 1.8)

A

Part Name -
Moving TableRev2

Friday, September 19, 2003 10:21:00 AM

Lesson Summary

- ❑ Base Feature is the first feature that is created — the foundation of the part.

- ❑ The Base Feature is the workpiece to which everything else is attached.

- ❑ You can create an Extruded Base Feature by selecting a sketch plane and extruding the sketch perpendicular to sketch plane.

- ❑ Shell Feature creates a hollow block from a solid block.

- ❑ The views most commonly used to describe a part are:
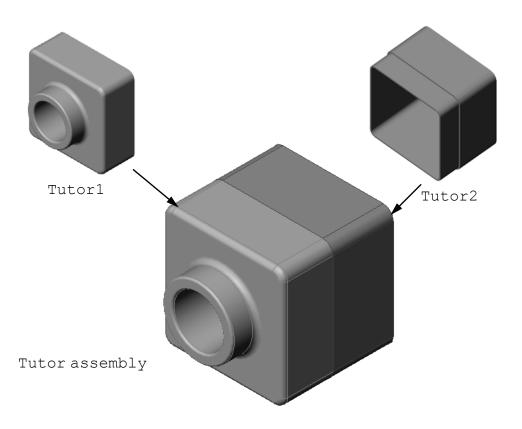 Top View
  Front View
  Right View
  Isometric View

4

Lesson 4: Assembly Basics

Goals of This Lesson

❑ Understand how parts and assemblies are related.

❑ Create and modify the part `Tutor2` and create the `Tutor` assembly.

`Tutor1`

`Tutor2`

`Tutor assembly`

Before Beginning This Lesson

❑ Complete the `tutor1` part in the previous lesson.
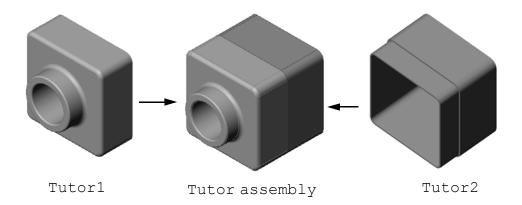
Resources for This Lesson

This lesson plan corresponds to *Lesson 2– Assemblies* in the SolidWorks Online Tutorials. For more information about the Online Tutorials, See "Online Tutorials" on page v.

Additional information about assemblies can be found in the *Assembly Mates* lesson in the SolidWorks Online Tutorials.

Active Learning Exercises — Creating an Assembly

Follow the instructions in *Lesson 2– Assemblies* in the SolidWorks Online Tutorials. In this lesson you will first create `Tutor2`. Then you will create an assembly.

`Tutor1`   `Tutor assembly`   `Tutor2`

5 Minute Assessment

1 What features did you use to create `Tutor2`?

_____

2 What two sketch tools did you use to create the extruded cut feature? _____

_____

3 What does the **Convert Entities** sketch tool do?_____

_____

4 What does the **Offset Entities** sketch tool do? _____

_____

5 In an assembly, parts are referred to as_____ .

_____

6 True or False. A fixed component is free to move. _____

_____

7 True or False. Mates are relationships that align and fit components together in an assembly. _____

_____

8 How many components does an assembly contain?_____

_____

9 What mates are required for the `Tutor assembly`? _____

_____

Exercises and Projects — Creating the Switchplate Assembly

Task 1— Modifying Feature Size

The `switchplate` created in Lesson 3 requires two fasteners to complete the assembly.

Question:

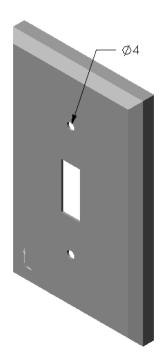How do you determine the size of the holes in the `switchplate`?

Answer:

_____

_____

_____

_____

_____

_____

Given:

❑ The diameter of the fastener is **3.5mm**.

❑ The switchplate is **10mm** deep.

Procedure:

1 **Open** the `switchplate`.
2 Modify the diameter of the two holes to **4mm**.
3 **Save** the changes.

Task 2 — Designing a Fastener

Design and model a fastener that is appropriate for the `switchplate`. Your fastener may (or may not) look like the one shown at the right.
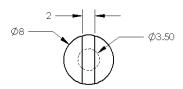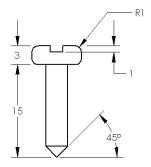
Design Criteria:

❑ The fastener must be longer than the thickness of the switchplate.

❑ The `switchplate` is **10mm** thick.

❑ The fastener must be **3.5mm** in diameter.

❑ The head of the fastener must be larger than the hole in the `switchplate`.

Good Modeling Practice

Fasteners are almost always modeled in a simplified form. That is, although a real machine screw has threads on it, these are not included in the model.

Task 3 — Creating an Assembly

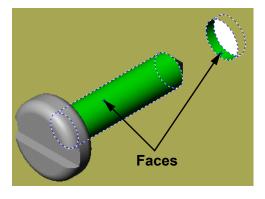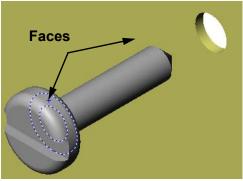Create the `switchplate-fastener` assembly.

Procedure:

1 Create a new assembly.
The fixed component is the `switchplate`.

2 Drag the `switchplate` into the assembly window.

3 Drag the `fastener` into the assembly window.

4 Use **Move Component** to position the `fastener` in front of the first hole.

The `switchplate-fastener` requires three mates to fully define the assembly.

5 Create a **Concentric** mate between the cylindrical face of the `fastener` and the cylindrical face of the hole in the `switchplate`.
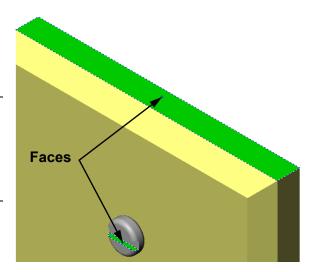

Faces

6 Create a **Coincident** mate between the back flat face of the `fastener` and the flat front face of the `switchplate`.

Faces

7 Create a **Parallel** mate between one of the flat faces on the slot of the `fastener` and the flat top face of the `switchplate`.

Note: If the necessary faces do not exist in the `fastener` or the `switchplate`, create the parallel mate using the appropriate reference planes in each component.
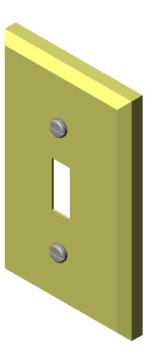

Faces

8 Add a second instance of the `fastener` to the assembly.

You can add components to an assembly by dragging and dropping:

- Hold the **Ctrl** key, and then drag the component either from the FeatureManager design tree, or from the graphics area.

- The pointer changes to ⌖.

- Drop the component in the graphics area by releasing the left mouse button and the **Ctrl** key.

9 Add three **mates** to fully define the second `fastener` to the `switchplate-fastener` assembly.

10 Save the `switchplate-fastener` assembly.

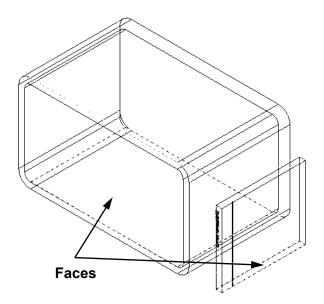## Exercises and Projects:— Creating CD Storage Box Assembly

Assemble the `cdcase` and `storagebox` that you created in Lesson 3.

Note: The completed `cdcase-storagebox` assembly example is found in the `Lesson3` file folder.
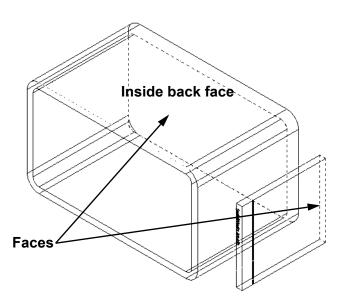
Procedure:

1 Create a new assembly.

   The fixed component is the `storagebox`.

2 Drag the `storagebox` into the assembly window.

   Locate the storagebox at the assembly origin using inferencing.

3 Drag the `cdcase` into the assembly window to the right of the `storagebox`.

4 Create a **Coincident** mate between the bottom face of the `cdcase` and the inside bottom face of the `storagebox`.

Faces

5 Create a **Coincident** mate between the back face of the `cdcase` and the inside back face of the `storagebox`.

Inside back face

Faces

6 Create a **Distance** mate between the *left* face of the cdcase and the inside left face of the storagebox.

Enter **1cm** for **Distance**.

7 Save the assembly.
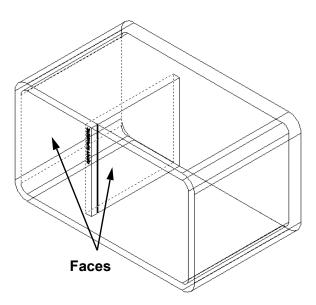
Enter cdcase-storagebox for the filename.
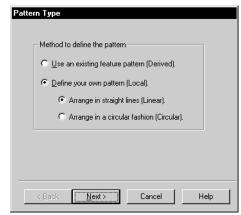
Component Patterns

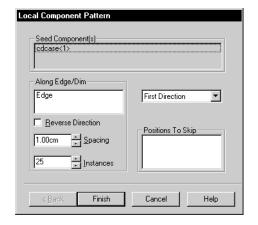Create a linear pattern of the cdcase component in the assembly.

The cdcase is the seed component. The seed component is what gets copied in the pattern.

1 Click **Insert**, **Component Pattern.**

The **Pattern Type** dialog is displayed.

2 Click **Define your own pattern (Local)**.

Make sure the option **Arrange in straight lines (Linear)** is selected.

3 Click **Next.**

The **Local Component Pattern** dialog box is displayed.

4 Select the component to be patterned.

Make sure the **Seed Component(s)** field is active, and then select the cdcase component from the FeatureManager design tree or the graphics area.

Faces

5   Define the direction for the pattern.

Click inside the **Along Edge/Dim** text box to make it active.

Click the top horizontal front edge of the `storagebox`.
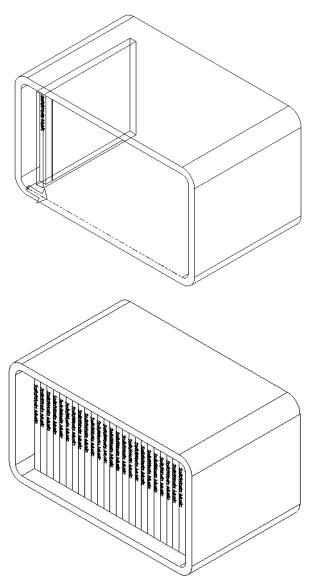
6   Observe the direction arrow.

The preview arrow should point to the right. If it does not, click the **Reverse Direction** check box.

7   Enter **2** for **Spacing**. Enter **25** for **Instances**. Click **Finish**.

The Local Component Pattern feature is added to the FeatureManager design tree.

8   Save the assembly.

Click **Save**. Use the name `cdcase-storagebox`.

Exercises and Projects: — Assembling a Mechanical Claw

Assemble the claw mechanism shown at the right. This assembly will be used later, in Lesson 11, to create a movie using the SolidWorks Animator software.

Procedure:

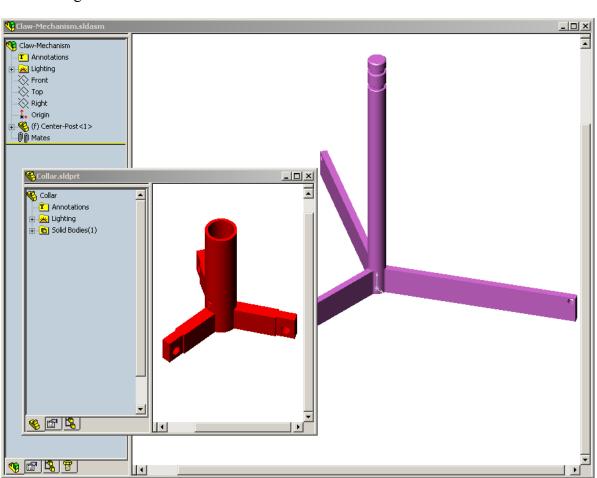**1** Create a new assembly.

2 Save the assembly. Name it `Claw-Mechanism`.

3 Insert the `Center-Post` component into the assembly.

The files for this exercises are found in the `Claw` folder in the `Lesson04` folder.

Position the `Center-Post` at the assembly origin. Make sure it is fully constrained.

4 Open the `Center-Post` part.
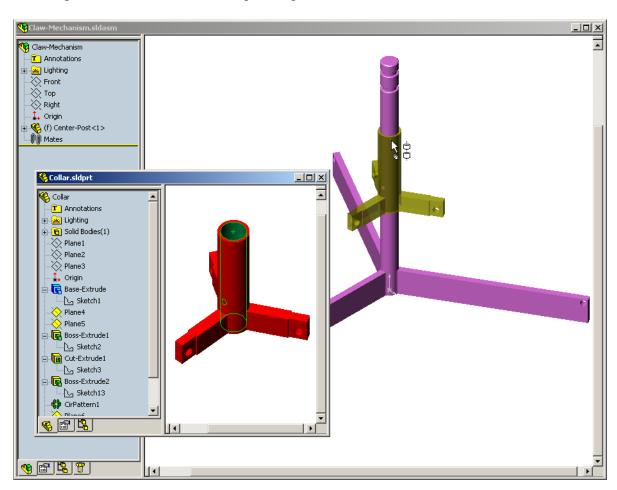
Arrange the windows as shown below.

SmartMates

You can create some types of mating relationships automatically. Mates created with these methods are referred to as SmartMates.

You can create mates when you drag the part in specific ways from an open part window. The entity that you use to drag determines the types of mates that are added.

5 Select the cylindrical face of the `Collar`, and drag the `Collar` into the assembly. Point at the cylindrical face of the `Center-Post` in the assembly window.

When the pointer is over the `Center-Post`, the pointer changes to ⊾⊟. This pointer indicates that a **Concentric** mate will result if the `Collar` is dropped at this location. A preview of the `Collar` snaps into place.
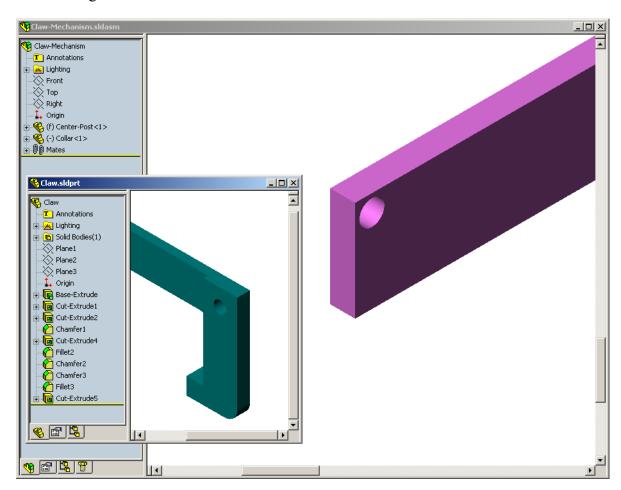

6 Drop the `Collar`.
A **Concentric** mate is added automatically.

7 Close the `Collar` part document.

8 Open the `Claw`.

Arrange the windows as shown below.

9 Add the `Claw` to the assembly using SmartMates

- Select the *edge* of the hole in the `Claw`.

It is important to select the edge and not the cylindrical face. This is because this type of SmartMate will add two mates:

- A **Concentric** mate between the cylindrical faces of the two holes.
- A **Coincident** mate between the planar face of the `Claw` and the arm of the `Center-Post`.

10 Drag and drop the `Claw` onto the *edge* of the hole in the arm.

The pointer looks like this 🖱 indicating that a **Concentric** and a **Coincident** mate will be added automatically. This SmartMate technique is ideal for putting fasteners into holes.
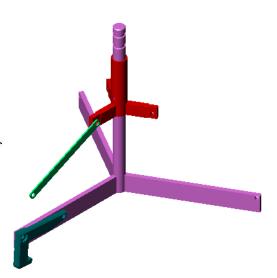
11 Close the `Claw` part document.

12 Add the `Connecting-Rod` to the assembly. Use the same SmartMate technique you used in steps 9 and 10 to mate one end of the `Connecting-Rod` to the end of the `Collar`.

There should be two mates:
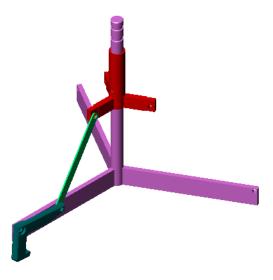
- **Concentric** between the cylindrical faces of the two holes.

- **Coincident** between the planar faces of the `Connecting-Rod` and the `Collar`.

13 Mate the `Connecting-Rod` to the `Claw`.

Add a **Concentric** mate between the hole in the `Connecting-Rod` and the hole in the `Claw`.

Do not add a **Coincident** mate between the `Connecting-Rod` and the `Claw`.

14 Add the pins.

There are three different length pins:

- `Pin-Long`

- `Pin-Medium`

- `Pin-Short`

Use the command **Tools, Measure** to determine which pin goes in which hole. Add the pins using SmartMates.

Circular Component Pattern

Create a circular pattern of the `Claw`, `Connecting-Rod`, and pins.

15 Click **Insert**, **Component Pattern.**

The **Pattern Type** dialog is displayed.

16 Click **Define your own pattern (Local)**.

Select the option **Arrange in a circular fashion (Circular)**.

17 Click **Next.**

The **Local Component Pattern** dialog box is displayed.

18 Select the components to be patterned.

Make sure the **Seed Component(s)** field is active, and then select the `Claw`, the `Connecting-Rod`, and the three pins.

19 Set the **Spacing** to 120°.

20 Set the **Instances** to 3.

21 Click **View, Temporary Axes**.

22 Click in the **Along Edge/Dim box**.
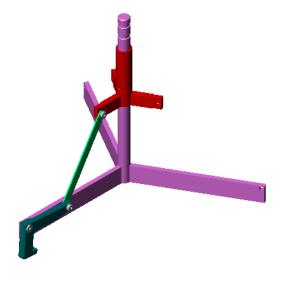
Select the axis that runs down the center of the `Center-Post` for the center of rotation for the pattern.
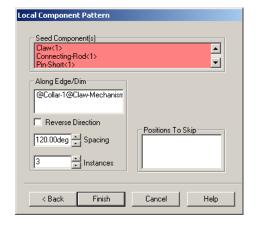
23 Click **Finish**.

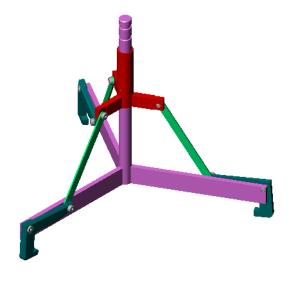**24** Turn off the temporary axes.

Dynamic Assembly Motion

Moving under defined components simulates movement of a mechanism through dynamic assembly motion.

25 Click **Move Component** .

Drag the `Collar` up and down while observing the motion of the assembly.

26 Save and close the assembly.

6
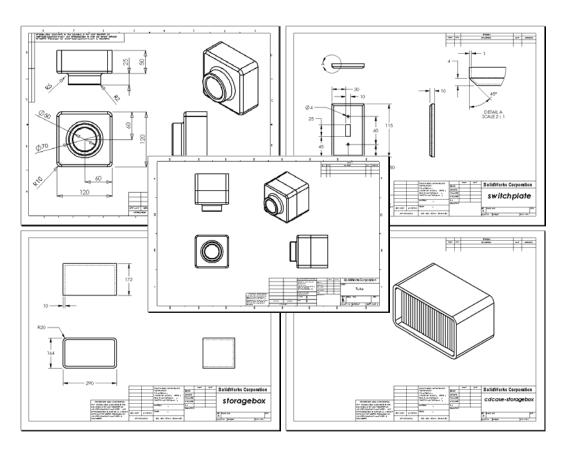
Lesson 6: Drawing Basics

Goals of This Lesson

❑ Understand basic drawing concepts.

❑ Create detailed drawings of parts and assemblies:.

Before Beginning This Lesson

❑ Create `Tutor1` and `Tutor2` parts and the `Tutor` assembly.

Resources for This Lesson
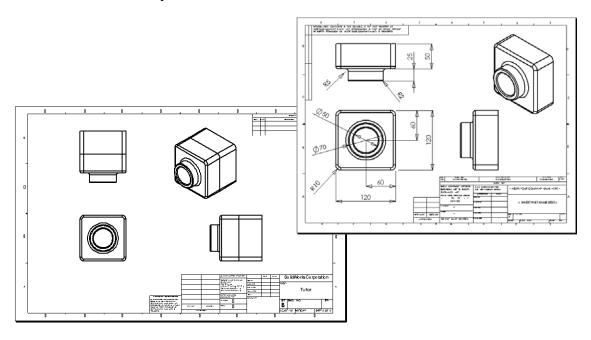
This lesson plan corresponds to *Lesson 3 – Drawings* in the SolidWorks Online Tutorials. For more information about the Online Tutorials, See "Online Tutorials" on page v.

Additional information about drawings can be found in the *Advanced Drawings* lesson and the *Bill of Materials* lesson in the SolidWorks Online Tutorials.

Active Learning Exercises — Creating Drawings

Follow the instructions in *Lesson 3 – Drawings* in the SolidWorks Online Tutorials. In this lesson you will create two drawings. First, you will create the drawing for the part named `Tutor1` which you built in a previous lesson. Then you will create an assembly drawing of the `Tutor` assembly.

5 Minute Assessment – Answer Key

1 How do you open a drawing template?

_____

_____

2 What is the difference between **Edit Sheet Format** and **Edit Sheet**?

_____

_____

_____

3 A title block contains information about the part and/or assembly. Name five pieces of information that can be contained in a title block.

_____

_____

_____

4 True or False. Right-click **Edit Sheet Format** to modify title block information.

_____

5 What three views are inserted into a drawing when you click **Standard 3 View**?

_____

_____

Exercises and Projects

Task 1— Create a Drawing Template

Create a new A-size ANSI standard drawing template.

For **Units** use millimeters.

Name the template `ANSI-MM-SIZEA`.

Procedure:

1 Create a new drawing using the Tutorial drawing template.
This is an A-size sheet that uses the ISO dimensioning standard.

2 Click **Tools, Options** and then click the **Document Properties** tab.

3 Click **Detailing** and set the **Dimensioning standard** to **ANSI**.

4 Make any other desired changes to the document properties, such as the dimension text font and size.

5 Click **Units** and verify that the units are set to millimeters.

6 Click **OK** to apply the changes and close the dialog.

7 Click **File, Save As...**

8 From the **Save as type:** list, click **Drawing Templates (*.drwdot)**.
The system automatically jumps to the directory where the templates are installed.

9 Click to create a new folder.

10 Name the new folder `Custom`.

11 Browse to the `Custom` folder.

12 Enter `ANSI-MM-SIZEA` for the name.

13 Click **Save**.
Drawing templates have the suffix `*.drwdot`
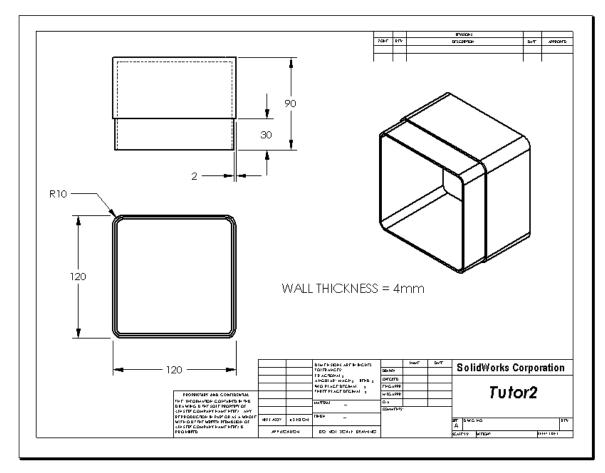
Task 2— Create a Drawing for Tutor2

1 Create a drawing for `Tutor2`. Use the drawing template you created in Task 1.
Review the guidelines for determining which views are necessary. Since `Tutor2` is square, the top and right views communicate the same information. Only two views are necessary to fully describe the shape of `Tutor2`.
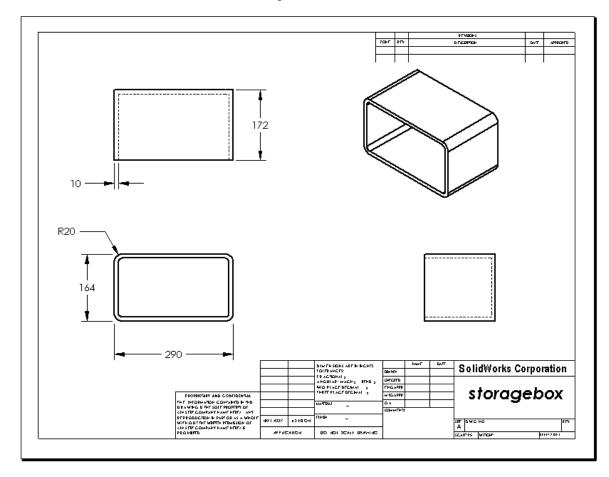
2 Create Front and Top views. Add an Isometric view.

3 Import the dimensions from the part.

4 Create a note on the drawing to label the wall thickness.
Right-click **Drawing Annotations, Note**. Enter **WALL THICKNESS = 4MM**.

Task 3— Add a Sheet to an Existing Drawing

1 Add a new sheet to the existing drawing you created in Task 2. Use the drawing template you created in Task 1.

2 Create a three standard views for the `storagebox`.

3 Import the dimensions from the model.

4 Create an Isometric view in a drawing for the `storagebox`.

Task 4— Add a Sheet to an Existing Assembly Drawing

1 Add a new sheet to the existing drawing you created in Task 2. Use the drawing template you created in Task 1.

2 Create an Isometric view in a drawing for the `cdcase-storagebox` assembly.
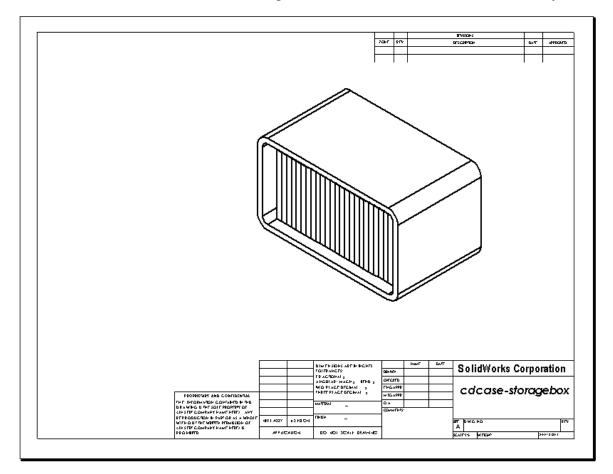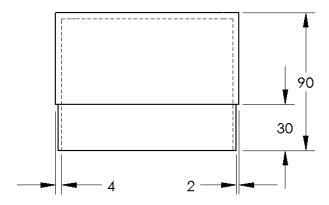
More to Explore — Create a Parametric Note

Investigate the on-line documentation to learn how to create a *parametric* note. In a parametric note, text, such as the numeric value of the wall thickness, is replaced with a dimension. This causes the note to update whenever the thickness of the shell is changed.

Once a dimension is linked to a parametric note, the dimension should *not* be deleted. That would break the link. However, the dimension can be hidden by right-clicking the dimension, and selecting **Hide** from the shortcut menu.

Procedure:

1 Import the model dimensions into the drawing.

When you import the dimensions from the model, the 4mm thickness dimension of the Shell feature will also be imported. This dimension is needed for the parametric note.

2 Click A or **Insert, Annotations, Note**.

TIP: To insert a note, you can also right-click in the graphics area, and select **Annotations, Note** from the shortcut menu.

---

3 Click to place the note on the drawing.

A text insertion box appears | ▢ |. Enter the note text. For example: **WALL THICKNESS =**

4 Select the dimension of the Shell feature.

Instead of typing the value, click the dimension. The system will enter the name of the dimension into the text note.

WALL THICKNESS = "D1 @Shell1 @Tutor2-2@Drawing View2"

5 Type the rest of the note.
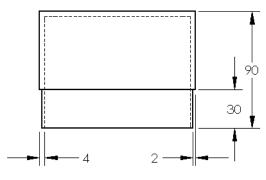
Make sure the text insertion cursor is at the end of the text string and type **mm**.

WALL THICKNESS = "D1 @Shell1 @Tutor2-2@Drawing View2"mm

6 Click **OK** to close the **Note** PropertyManager.

Position the note on the drawing by dragging it.

7 Hide the dimension.

Right-click the dimension, and select **Hide** from the shortcut menu.

You should *not* delete the dimension that was referenced in the parametric note. If you do, a change made to that dimension in the model will not propagate to the note. Instead you should hide the dimension.

More to Explore — Add a Sheet to Switchplate Drawing
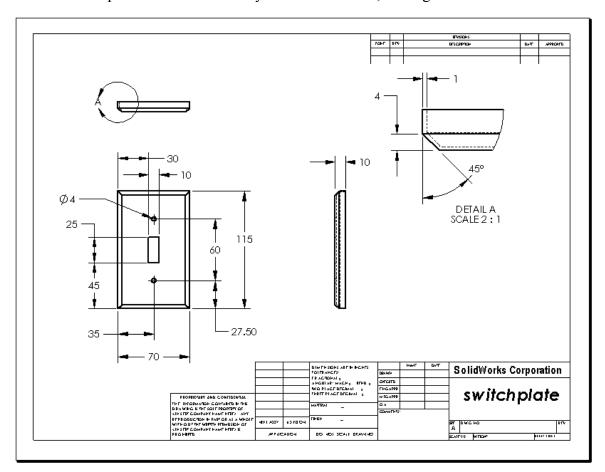
1 Add a new sheet to the existing drawing you created in Task 2. Use the drawing template you created in Task 1.

2 Create a drawing of the `switchplate`.

The chamfer is too small to be clearly seen and dimensioned in either the Top or Right views. A detail view is required. Detail views are views that usually show only a portion of the model, at a larger scale. To make a detail view:

3 Select the view from which the detail view will be derived.

4 Click **Detail View** [icon], or **Insert, Drawing View, Detail**.

This turns on the Circle sketch tool.

5 Sketch a circle around the area you want to show.

When you finish sketching the circle, a preview of the detail view appears.

6 Position the detail view on the drawing sheet.

The system automatically adds a label to the detail circle and the view itself. To change the scale of the detail view, edit the label's text.

7 You can import dimensions directly into a detail view, or drag them from other views.

Lesson Summary
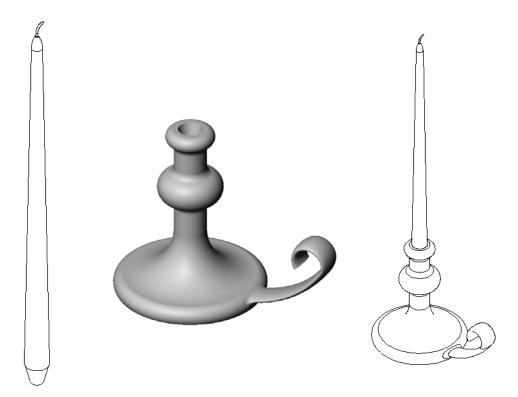
❑ Engineering Drawings communicate three things about the objects they represent:

- Shape – *Views* communicate the shape of an object.

- Size – *Dimensions* communicate the size of an object.

- Other information – *Notes* communicate non-graphic information about manufacturing processes such as drill, ream, bore, paint, plate, grind, heat treat, remove burrs, and so forth.

❑ The general characteristics of an object will determine what views are required to describe its shape.

❑ Most objects can be described using three properly selected views.

❑ There are two kinds of dimensions:

- Size dimensions – how big is the feature?

- Location dimensions – where is the feature?

❑ A drawing template specifies:

- Sheet (paper) size

- Orientation - Landscape or Portrait

- Sheet Format

Lesson 9: Revolve and Sweep Features

Goals of This Lesson

Upon successful completion of this lesson, you 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.

Active Learning Exercises — Creating a Candlestick

Create the candlestick. Follow the instructions in the *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 – Answer Key

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

_____

_____

_____

2 What special piece of sketch geometry is *required* for a revolve feature?

_____

_____

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

_____

_____

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

_____

_____

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

_____

Why?_____

_____

_____

_____

_____

A   **B**   **C**

Exercises and Projects — Creating a Candle to Fit the Candlestick

Task 1— Revolve Feature
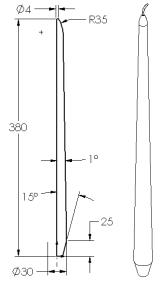
Design a candle to fit the candlestick.

❑ 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:

_____

_____

_____

_____

_____

_____

_____

_____

_____

Question:

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

Answer:

_____

_____

_____

_____

_____

_____

_____

_____

_____

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 15 inch, 12 inch, 10 inch and 7 inch candles.

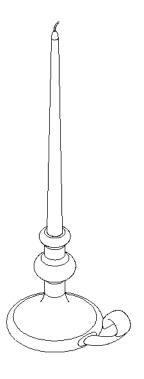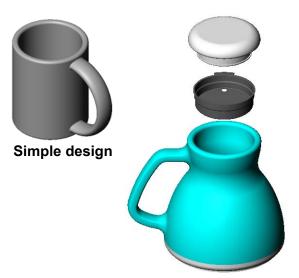|	A	B
1	Design Table for: candle	
2		Length@Sketch1
3	15 inch candle	15
4	12 inch candle	12
5	10 inch candle	10
6	7 inch candle	7

Sheet1

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.

Simple design
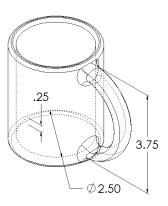
More complex design – a commuter's spill-proof travel mug

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.

.25

3.75

⌀ 2.50

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 in^3
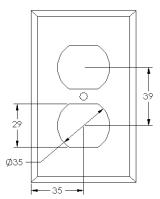
128 ounces = 1 gallon

Answer:

_____

_____

_____

_____

_____

_____

_____

Exercises and Projects — Modify the Outlet Plate
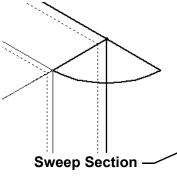
Modify the `outletplate` that you created in earlier.

❑ 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.
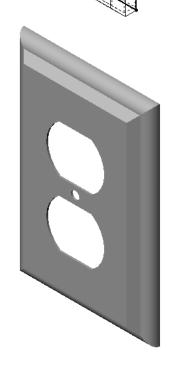
❑ Add a swept boss feature to the back edge.

- 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.
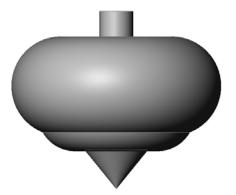
- Use geometric relations to fully define the sweep section sketch.

- The sweep path is made up of the four rear edges of the part.

- Use **Convert Entities** to create the sweep path.
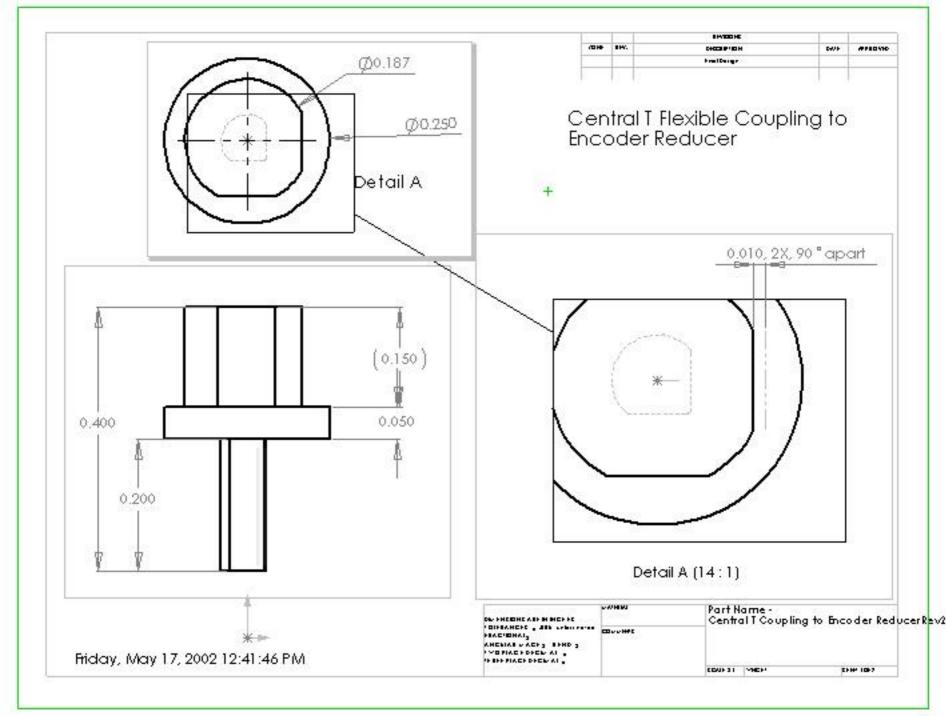
❑ The desired result is shown in the illustration at the right.

More to Explore — Use Revolve Feature to Design a Top
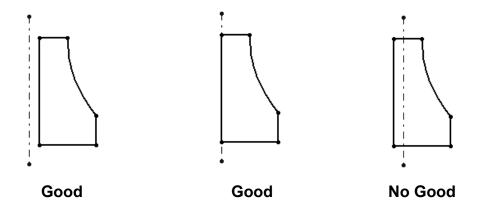
Use a revolve feature to create a toy top of your own design.

Ø0.187

Ø0.250

Detail A

Central T Flexible Coupling to
Encoder Reducer

0.010, 2X, 90° apart

(0.150)

0.400

0.050

0.200

Detail A (14 : 1)

Friday, May 17, 2002 12:41:46 PM

Lesson Summary

❑ A Revolve feature is created by rotating a 2D profile sketch around a centerline.

❑ The profile sketch *must* contain the centerline.

❑ The profile sketch *cannot* cross the centerline.

Good   **Good**   **No Good**
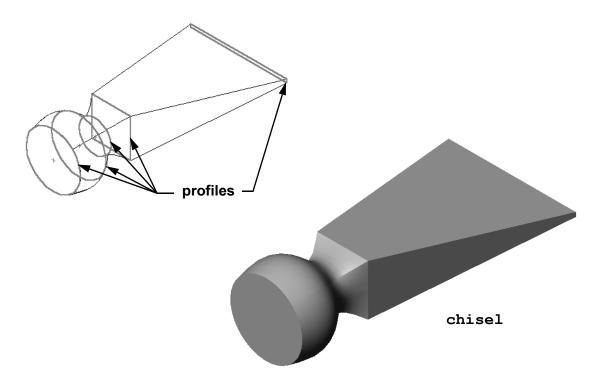
❑ The Sweep feature is created by moving a 2D profile along a path.

❑ 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.

10

Lesson 10: Loft Features

Goals of This Lesson

❑ You will be able to create the following part:

profiles

chisel

Resources for This Lesson
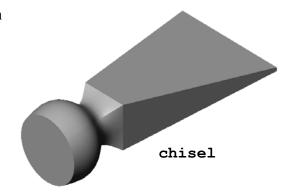
This lesson plan corresponds to the *Lofts* module in the SolidWorks Online Tutorials. For more information about the Online Tutorials, See "Online Tutorials" on page v.

Active Learning Exercises — Creating the Chisel

Create the `chisel`. Follow the instructions in the *Lofts* module in the SolidWorks Online Tutorials.

chisel
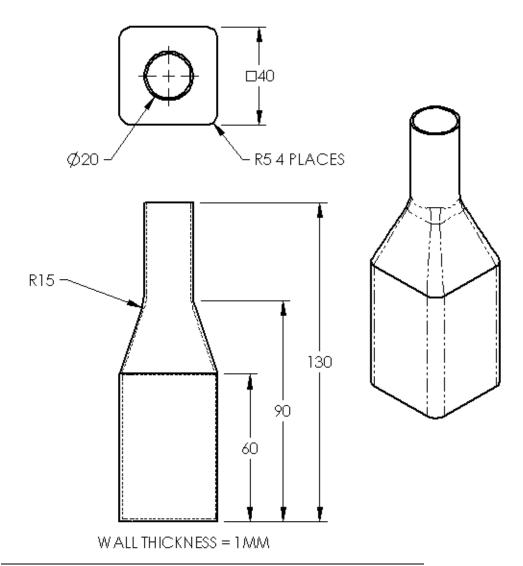
5 Minute Assessment – Answer Key

1 What features were used to create the `chisel`? _____

_____

_____

_____

2 Describe the steps required to create the first Loft feature for the `chisel`.

_____

_____

_____

_____

_____

_____

_____

_____

3 What is the minimum number of profiles required for a Loft feature? _____

_____

4 Describe the steps to copy a Sketch onto another plane.

_____

_____

_____

_____

_____

_____

_____

_____

_____

Exercises and Projects — Creating the Bottle

Create the `bottle` as shown in the drawing.

□40

Ø20

R5 4 PLACES

R15

130

90

60

WALL THICKNESS = 1MM
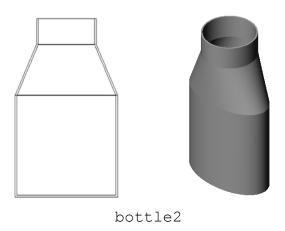
Note: All dimensions in the Bottle exercise are in millimeters.

Exercises and Projects — Creating a Bottle with Elliptical Base
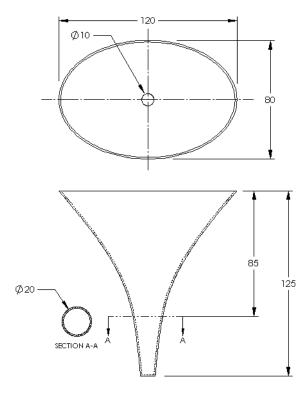
Create `bottle2` with an elliptical extruded boss feature. The top of the bottle is circular. Design `bottle2` with your own dimensions.

bottle2

Exercises and Projects — Creating a Funnel

Create the `funnel` as shown in the drawing below.

❑ Use **1mm** for the wall thickness.
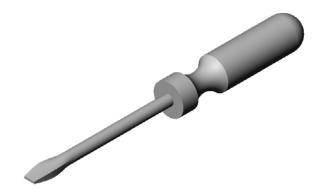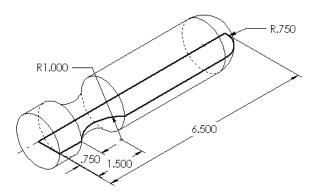
Exercises and Projects — Creating a Screwdriver

Create the `screwdriver`.
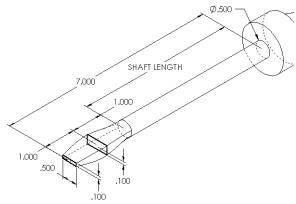
❑ Use **inches** for the units.
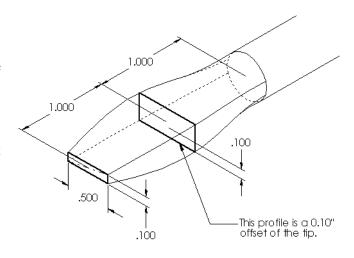
❑ Create the handle as the first feature.
   Use a revolved feature.

❑ Create the shaft as the second feature.
   Use an extruded feature.

❑ The overall length of the blade (shaft and
   tip together) is **7 inches**. The tip is **2
   inches** long. Compute the length of the
   shaft.

❑ Create the tip as the third feature.
   Use a loft feature.

❑ Create the sketch for the end of the
   tip first. This is a rectangle **0.50"**
   by **0.10"**.

❑ The middle — or second profile
   — is sketched using a **0.10"** offset
   (to the outside) of the tip.

❑ The third profile is the circular
   face on the end of the shaft.
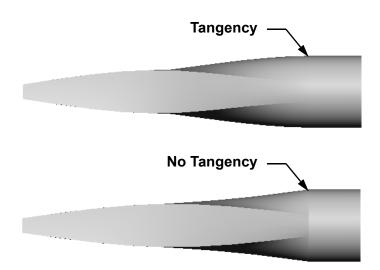
Matching Tangency

When you want to blend a loft feature into an existing feature such as the shaft, it is desirable to have the face blend smoothly.
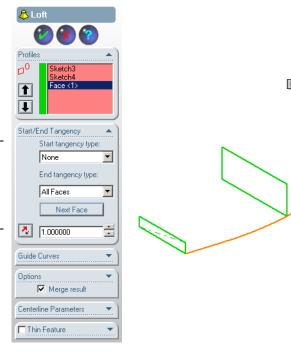
Look at the illustrations at the right. In the upper one, the tip was lofted with tangency matching to the shaft. The lower example was not.

Tangency

No Tangency

In the **Start/End Tangency** box of the PropertyManager, there are some tangency options. **End tangency** applies to the last profile, which in this case, is the face on the end of the shaft.

Note: If you picked the face of the shaft as the *first* profile, you would use the **Start tangency** option.

The option **All faces** will make the lofted feature tangent to the sides of the shaft.

The result is shown at the right.

More to Explore — Designing a Sports Drink Bottle

Task 1

❑ Design a 16 ounce `sportsbottle`. How would you calculate the capacity of the bottle?

❑ Create a `cap` for the `sportsbottle`.

❑ Create a `sportsbottle` assembly.

Question

How many liters are contained in the `sportsbottle`?

Conversion

❑ 1 fluid ounce = 29.57ml

Answer:

_____

_____

_____

_____

_____

cap —

sports — bottle

sportsbottle assembly

Task 2

A designer for your company receives the following cost information:

❑ Sports Drink = $0.32 per gallon based on 10,000 gallons

❑ 16 ounce sport bottle = $0.11 each based on 50,000 units

Question

How much does it cost to produce a filled 16 oz. sportsbottle to the nearest cent?

Answer:

_____

_____

_____

_____

_____

_____

_____

Lesson Summary

❏ A Loft blends multiple profiles together.

❏ A Loft feature can be a base, boss, or cut.

❏ Neatness counts!

  • Select the profiles in order.

  • Click corresponding points on each profile.

  • The vertex closest to the selection point is used.

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Glossary

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animate	View a model or eDrawing in a dynamic manner. Animation simulates motion or displays different views.
assembly	An assembly is a document in which parts, features, and other assemblies (sub-assemblies) are mated together. The parts and sub-assemblies exist in documents separate from the assembly. For example, in an assembly, a piston can be mated to other parts, such as a connecting rod or cylinder. This new assembly can then be used as a sub-assembly in an assembly of an engine. The extension for a SolidWorks assembly file name is.SLDASM. See also sub-assembly and mate.
axis	An axis is a straight line that can be used to create model geometry, features, or patterns. An axis can be made in a number of different ways, including using the intersection of two planes. See also temporary axis, reference geometry
block	A block is a user-defined annotation for drawings only. A block can contain text, sketch entities (except points), and area hatch, and it can be saved in a file for later use as, for example, a custom callout or a company logo.
boss/base	A base is the first solid feature of a part, created by a boss. A boss is a feature that creates the base of a part, or adds material to a part, by extruding, revolving, sweeping, or lofting a sketch, or by thickening a surface.
broken-out section	A broken-out section exposes inner details of a drawing view by removing material from a closed profile, usually a spline.
chamfer	A chamfer bevels a selected edge or vertex.
click-click	As you sketch, if you click and then release the pointer, you are in click-click mode. Move the pointer and click again to define the next point in the sketch sequence.
click-drag	As you sketch, if you click and drag the pointer, you are in click-drag mode. When you release the pointer, the sketch entity is complete.

closed profile	A closed profile (or closed contour) is a sketch or sketch entity with no exposed endpoints; for example, a circle or polygon.
collapse	Collapse is the opposite of explode. The collapse action returns an exploded assembly's parts to their normal positions.
component	A component is any part or sub-assembly within an assembly.
configuration	A configuration is a variation of a part or assembly within a single document. Variations can include different dimensions, features, and properties. For example, a single part such as a bolt can contain different configurations that vary the diameter and length. See design table.
Configuration Manager	The ConfigurationManager on the left side of the SolidWorks window is a means to create, select, and view the configurations of parts and assemblies.
coordinate system	A coordinate system is a system of planes used to assign Cartesian coordinates to features, parts, and assemblies. Part and assembly documents contain default coordinate systems; other coordinate systems can be defined with reference geometry. Coordinate systems can be used with measurement tools and for exporting documents to other file formats.
degrees of freedom	Geometry that is not defined by dimensions or relations is free to move. In 2D sketches, there are three degrees of freedom: movement along the X and Y axes, and rotation about the Z axis (the axis normal to the sketch plane). In 3D sketches and in assemblies, there are six degrees of freedom: movement along the X, Y, and Z axes, and rotation about the X, Y, and Z axes. See under defined.
design table	A design table is an Excel spreadsheet that is used to create multiple configurations in a part or assembly document. See configurations.
document	A SolidWorks document is a file containing a part, assembly, or drawing.
drawing	A drawing is a 2D representation of a 3D part or assembly. The extension for a SolidWorks drawing file name is.SLDDRW.
drawing sheet	A drawing sheet is a page in a drawing document.
eDrawing	Compact representation of a part, assembly, or drawing. eDrawings are compact enough to email and can be created for a number of CAD file types including SolidWorks.
face	A face is a selectable area (planar or otherwise) of a model or surface with boundaries that help define the shape of the model or surface. For example, a rectangular solid has six faces. See also surface.

feature	A feature is an individual shape that, combined with other features, makes up a part or assembly. Some features, such as bosses and cuts, originate as sketches. Other features, such as shells and fillets, modify a feature's geometry. However, not all features have associated geometry. Features are always listed in the FeatureManager design tree. See also surface, out-of-context feature.
FeatureManager design tree	The FeatureManager design tree on the left side of the SolidWorks window provides an outline view of the active part, assembly, or drawing.
fillet	A fillet is an internal rounding of a corner or edge in a sketch, or an edge on a surface or solid.
graphics area	The graphics area is the area in the SolidWorks window where the part, assembly, or drawing appears.
helix	A helix is defined by pitch, revolutions, and height. A helix can be used, for example, as a path for a swept feature cutting threads in a bolt.
instance	An instance is an item in a pattern or a component that occurs more than once in an assembly.
layer	A layer in a drawing can contain dimensions, annotations, geometry, and components. You can toggle the visibility of individual layers to simplify a drawing or assign properties to all entities in a given layer.
line	A line is a straight sketch entity with two endpoints. A line can be created by projecting an external entity such as an edge, plane, axis, or sketch curve into the sketch.
loft	A loft is a base, boss, cut, or surface feature created by transitions between profiles.
mate	A mate is a geometric relationship, such as coincident, perpendicular, tangent, and so on, between parts in an assembly. See also SmartMates.
mategroup	A mategroup is a collection of mates that are solved together. The order in which the mates appear within the mategroup does not matter.
mirror	(1) A mirror feature is a copy of a selected feature, mirrored about a plane or planar face. (2) A mirror sketch entity is a copy of a selected sketch entity that is mirrored about a centerline. If the original feature or sketch is modified, the mirrored copy is updated to reflect the change.

model   A model is the 3D solid geometry in a part or assembly document. If a part or assembly document contains multiple configurations, each configuration is a separate model.

mold   A mold cavity design requires (1) a designed part, (2) a mold base that holds the cavity for the part, (3) an interim assembly in which the cavity is created, and (4) derived component parts that become the halves of the mold.

named view   A named view is a specific view of a part or assembly (isometric, top, and so on) or a user-defined name for a specific view. Named views from the view orientation list can be inserted into drawings.

open profile   An open profile (or open contour) is a sketch or sketch entity with endpoints exposed. For example, a U-shaped profile is open.

origin   The model origin appears as three gray arrows and represents the (0,0,0) coordinate of the model. When a sketch is active, a sketch origin appears in red and represents the (0,0,0) coordinate of the sketch. Dimensions and relations can be added to the model origin, but not to a sketch origin.

over defined   A sketch is over defined when dimensions or relations are either in conflict or redundant.

parameter   A parameter is a value used to define a sketch or feature (often a dimension).

part   A part is a single 3D object made up of features. A part can become a component in an assembly, and it can be represented in 2D in a drawing. Examples of parts are bolt, pin, plate, and so on. The extension for a SolidWorks part file name is .SLDPRT.

pattern   A pattern repeats selected sketch entities, features, or components in an array, which can be linear, circular, or sketch-driven. If the seed entity is changed, the other instances in the pattern update.

planar   An entity is planar if it can lie on one plane. For example, a circle is planar, but a helix is not.

plane   Planes are flat construction geometry. Planes can be used for a 2D sketch, section view of a model, a neutral plane in a draft feature, and others.

point   A point is a singular location in a sketch, or a projection into a sketch at a single location of an external entity (origin, vertex, axis, or point in an external sketch). See also vertex.

profile	A profile is a sketch entity used to create a feature (such as a loft) or a drawing view (such as a detail view). A profile can be open (such as a U shape or open spline) or closed (such as a circle or closed spline).
Property Manager	The PropertyManager is on the left side of the SolidWorks window for dynamic editing of sketch entities and most features.
rebuild	The rebuild tool updates (or regenerates) the document with any changes made since the last time the model was rebuilt. Rebuild is typically used after changing a model dimension.
relation	A relation is a geometric constraint between sketch entities or between a sketch entity and a plane, axis, edge, or vertex. Relations can be added automatically or manually.
revolve	Revolve is a feature tool that creates a base or boss, a revolved cut, or revolved surface by revolving one or more sketched profiles around a centerline.
section	A section is another term for profile in sweeps.
section view	A section view (or section cut) is (1) a part or assembly view cut by a plane, or (2) a drawing view created by cutting another drawing view with a section line.
shaded	A shaded view displays a model as a colored solid. See also HLR, HLG, and wireframe.
sheet format	A sheet format typically includes page size and orientation, standard text, borders, title blocks, and so on. Sheet formats can be customized and saved for future use. Each sheet of a drawing document can have a different format.
shell	Shell is a feature tool that hollows out a part, leaving open the selected faces and thin walls on the remaining faces. A hollow part is created when no faces are selected to be open.
sketch	A 2D sketch is a collection of lines and other 2D objects on a plane or face that forms the basis for a feature such as a base or a boss. A 3D sketch is non-planar and can be used to guide a sweep or loft, for example.
SmartMates	A SmartMate is an assembly mating relation that is created automatically. See mate.
sub-assembly	A sub-assembly is an assembly document that is part of a larger assembly. For example, the steering mechanism of a car is a sub-assembly of the car.

surface   A surface is a zero-thickness planar or 3D entity with edge boundaries. Surfaces are often used to create solid features. Reference surfaces can be used to modify solid features. See also face.

sweep   A sweep creates a base, boss, cut, or surface feature by moving a profile (section) along a path.

template   A template is a document (part, assembly, or drawing) that forms the basis of a new document. It can include user-defined parameters, annotations, or geometry.

toolbox   A library of standard parts that are fully integrated with SolidWorks. These parts are ready-to-use components — such as bolts and screws.

under defined   A sketch is under defined when there are not enough dimensions and relations to prevent entities from moving or changing size. See degrees of freedom.

vertex   A vertex is a point at which two or more lines or edges intersect. Vertices can be selected for sketching, dimensioning, and many other operations.

wireframe   Wireframe is a view mode in which all edges of the part or assembly are displayed. See also HLR, HLG, shaded.