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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 appear 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:

- **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 not 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.
Lesson 1: Using the Interface

Goals of This Lesson

- Become familiar with the Microsoft Windows interface.
- Become familiar with the SolidWorks interface.

Before Beginning This Lesson

- Verify that Microsoft Windows is loaded and running on your computer.
- Verify that the SolidWorks software is loaded and running on your computer in accordance with your SolidWorks license.
- Verify that the template and lesson files from the Companion Files CD have been loaded on your computer.

Resources for This Lesson

- Introducing SolidWorks, Chapter 1.
Active Learning Exercise — Using the Interface

Start the SolidWorks application, search for a file, save the file, save the file with a new name, and review the basic user interface.

The step-by-step instructions are given below.

Starting a Program

1. Click the **Start** button in the lower left corner of the window. The **Start** menu appears. The **Start** menu allows you to select the basic functions of the Microsoft Windows environment.

   **Note:** Click means to press and release the left mouse button.

2. From the **Start** menu, click **Programs, SolidWorks, SolidWorks** as shown below. The SolidWorks application program is now running.

   **Note:** Your **Start** menu may appear different than the illustration depending on which versions of software are loaded on your system.

   **TIP:** A desktop shortcut is an icon that you can double-click to go directly to the file or folder represented. If your system desktop has a shortcut to the SolidWorks application program, you can start the program by double-clicking the left mouse button on this shortcut. The illustration shows the SolidWorks shortcut.
Exit the Program

To exit the application program, click **File, Exit** or click on the main SolidWorks window.

Searching for a File or Folder

You can search for files (or folders containing files). This is useful if you cannot remember the exact name of the file that you need.

3 Click **Start, Search**. Search for the SolidWorks part dumbell. To do this, enter dumb* in the **Search for files of folders named** field. Specifying what to search for and where to search for it is known as defining the search criteria.

**TIP:** The asterisk (*) is a wild card. The wild card allows you to enter part of a file name and search for all files and folders that contain that piece.

4 Click **Search Now**.

The files and folders that match the search criteria appear in the **Search Results** window.

**TIP:** You can also begin a search by right-clicking on the **Start** button and selecting **Search**. Right-click means to press and release the right button on your mouse.

Opening an Existing File

5 Double-click on the SolidWorks part file Dumbell.

This opens the Dumbell file in SolidWorks. If the SolidWorks application program is not running when you double-click on the part file name, the system runs the SolidWorks application program and then opens the part file that you selected.

**TIP:** Use the left mouse button to double-click. Double-clicking with the left mouse button is often a quick way of opening files from a folder.

You could have also opened the file by selecting **Open, Open from Web Folder**, or a file name from the **File** menu in SolidWorks. SolidWorks lists the last several files that you had open.

Saving a File

6 Click  to save changes to a file.

It is a good idea to save the file that you are working whenever you make changes to it.
Copying a File

Notice that Dumbell is not spelled correctly. It is supposed to have two “b’s”.

1 Click File, Save As to save a copy of the file with a new name.

The Save As window appears. This window shows you in which folder the file is currently located, the file name, and the file type.

2 In the File Name field enter the name Dumbbell and click Save.

A new file is created with the new name. The original file still exists. The new file is an exact copy of the file as it exists at the moment that it is copied.

Resizing Windows

SolidWorks, like many applications, uses windows to show your work. You can change the size of each window.

1 Move the cursor along the edge of a window until the shape of the cursor appears to be a two-headed arrow.

2 While the cursor still appears to be a two-headed arrow, hold down the left mouse button and drag the window to a different size.

3 When the window appears to be the size that you wish, release the mouse button.

Windows can have multiple panels. You can resize these panels relative to each other.

4 Move the cursor along the border between two panels until the cursor appears to be two parallel lines with perpendicular arrows.

5 While the cursor still appears to be two parallel lines with perpendicular arrows, hold down the left mouse button and drag the panel to a different size.

6 When the panel appears to be the size that you wish, release the mouse button.

SolidWorks Windows

SolidWorks windows have two panels. One panel provides non-graphic data. The other panel provides graphic representation of the part, assembly, or drawing.

The leftmost panel of the window contains the FeatureManager® design tree, PropertyManager, ConfigurationManager, and Toolbox.

1 Click each of the tabs at the bottom of the left panel and see how the contents of the window changes.
The rightmost panel is the Graphics Area, where you create and manipulate the part, assembly, or drawing.

2 Look at the Graphics Area. See how the dumbbell is represented. It appears shaded, in color, in an isometric view, and with shadows. These are some of the ways in which the model can be represented very realistically.

Toolbars

Toolbar buttons are shortcuts for frequently used commands. You can set toolbar placement and visibility based on the document type (part, assembly, or drawing). SolidWorks remembers which toolbars to display and where to display them for each document type.

1 Click View, Toolbars.

A list of all toolbars displays. The toolbars with a check mark beside them are visible; the toolbars without a check mark are hidden.

2 Click the toolbar name to turn its display on or off. If it is not already on, click View to turn the View toolbar on.

3 Turn several toolbars on and off to see the commands.

Mouse Buttons

Mouse buttons operate in the following ways:

- **Left** – Selects menu items, entities in the graphics area, and objects in the FeatureManager design tree.
- **Right** – Displays the context-sensitive shortcut menus.
- **Middle** – Rotates, pans, and zooms the view of a part or an assembly, and pans in a drawing.
Shortcut Menus

Shortcut menus give you access to a wide variety of tools and commands while you work in SolidWorks. When you move the pointer over geometry in the model, over items in the FeatureManager design tree, or over the SolidWorks window borders, right-clicking pops up a shortcut menu of commands that are appropriate for wherever you clicked.

You can access the "more commands menu" by selecting the double-down arrows in the menu. When you select the double-down arrows or pause the pointer over the double-down arrows, the shortcut menu expands to offer more menu items.

The shortcut menu provides an efficient way to work without continually moving the pointer to the main pull-down menus or the toolbar buttons.

Getting Online Help

If you have questions while you are using the SolidWorks software, you can find answers in several ways.

Note: If the Help button does not appear in the Standard toolbar, you can add it. To do so, click Tools, Customize, Commands, and the toolbar that you wish to add the button to. In this case, click Standard. The available buttons for that toolbar display. Drag the button to the toolbar at the top of the SolidWorks window.

1. Click or Help, SolidWorks Help Topics in the menu bar.

   The online help appears.

2. Click on the Standard toolbar, then click a toolbar icon or a FeatureManager item.

   What’s This? help appears in a new window.
5 Minute Assessment

1. Search for the SolidWorks part file Paper Towel Base. How did you find it?

_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________

2. What is the quickest way to bring up the Search window?

_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________

3. How do you open the file from the Search Results window?

_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________

4. How do you start the SolidWorks program?

_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________

5. What is the quickest way to start the SolidWorks program?

_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________
Lesson 1 Vocabulary Worksheet

Name: _______________________________ Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided.

1  Shortcuts for collections of frequently used commands: ________________________  
   _______________________________________________________________________

2  Command to create a copy of a file with a new name: __________________________  
   _______________________________________________________________________

3  One of the areas that a window is divided into: _______________________________  
   _______________________________________________________________________

4  The graphic representation of a part, assembly, or drawing: ____________________  
   _______________________________________________________________________

5  Character that you can use to perform wild card searches: ______________________  
   _______________________________________________________________________

6  Area of the screen that displays the work of a program: _________________________  
   _______________________________________________________________________

7  Icon that you can double-click to start a program: _____________________________  
   _______________________________________________________________________

8  Action that quickly displays menus of frequently used or detailed commands: _______  
   _______________________________________________________________________
   _______________________________________________________________________

9  Command that updates your file with changes that you have made to it: __________  
   _______________________________________________________________________
   _______________________________________________________________________

10 Action that quickly opens a part or program: _________________________________  
   _______________________________________________________________________

11 The program that helps you create parts, assemblies, and drawings: ______________  
   _______________________________________________________________________

12 Panel of the SolidWorks window that displays a visual representation of your parts,  
   assemblies, and drawings: ________________________________________________  
   _______________________________________________________________________

13 Technique that allows you to find all files and folders that begin or end with a specified  
   set of characters: ___________________________________________________________________
Lesson Summary

- The Start menu is where you go to start programs or find files.
- You can use wild cards to search for files.
- There are shortcuts such as right-click and double-click that can save you work.
- **File, Save** allows you to save updates to a file and **File, Save As** allows you to make a copy of a file.
- You can change the size and location of windows as well as panels within windows.
- The SolidWorks window has a Graphics Area that shows 3D representations of your models.
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

2. Click the Tutorial tab.
3. Select the Part icon.

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 ![Sketch Tool] 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.

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 \( \Rightarrow \) 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 \( \Rightarrow \). 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**
- \( \Rightarrow \) (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 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.

12 Click OK.

**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.
Sketch the Circle

1. Click [Sketch Tools] 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 [Sketch Relations] 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 leftmost 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 bottommost horizontal edge. Click a location for the dimension text to the right of the sketch. Enter 40.

Extrude the Sketch

1. Click [Extrude] 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.
5 Minute Assessment

1. How do you start a SolidWorks session?

_____________________________________________________________________
_____________________________________________________________________

2. Why do you create and use Document Templates?

_____________________________________________________________________
_____________________________________________________________________

3. How do you start a new Part Document?

_____________________________________________________________________
_____________________________________________________________________

4. What features did you use to create the box?

_____________________________________________________________________
_____________________________________________________________________

5. True or False. SolidWorks is used by designers and engineers.

_____________________________________________________________________
_____________________________________________________________________

6. A SolidWorks 3D model consists of.

_____________________________________________________________________
_____________________________________________________________________

7. How do you open a sketch?

_____________________________________________________________________
_____________________________________________________________________

8. What does the Fillet feature do?

_____________________________________________________________________
_____________________________________________________________________

9. What does the Shell feature do?

_____________________________________________________________________
_____________________________________________________________________

10. What does the Cut-Extrude feature do?

_____________________________________________________________________
_____________________________________________________________________

11. How do you change a dimension value?

_____________________________________________________________________
_____________________________________________________________________
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 2 Vocabulary Worksheet

Fill in the blanks with the words that are defined by the clues. Then find the words in the puzzle and circle them. The words may be vertical, horizontal, or diagonal. They may be spelled forward or backward.

1. The corner or point where edges meet: ______________________________

2. The intersection of the three default reference planes: ____________________

3. A feature used to round off sharp corners: _______________________________

4. The three types of documents that make up a SolidWorks model: _______________

5. A feature used to hollow out a part: _________________________________

6. Controls the units, grid, text, and other settings of the document: _______________

7. Forms the basis of all extruded features: _______________________________

8. Two lines that are at right angles (90°) to each other are: ____________________

9. The first feature in a part is called the __________________ feature.

10. The outside surface or skin of a part: _________________________________

11. A mechanical design automation software application: ____________________

12. The boundary of a face: _________________________________

13. Two straight lines that are always the same distance apart are: _______________

14. Two circles or arcs that share the same center are: ______________________

15. The shapes and operations that are the building blocks of a part: _______________

16. A feature that adds material to a part: _________________________________

17. A feature that removes material from a part: ______________________________

18. An implied centerline that runs through the center of every cylindrical feature: ________
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.
Lesson 3: The 40-Minute Running Start

Goals of This Lesson

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

![Diagram of a part with views: 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.

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:

- $1\text{cm} = 10\text{mm}$

Answer:

_______________________________________________________________________
_______________________________________________________________________
_______________________________________________________________________
_______________________________________________________________________
_______________________________________________________________________
_______________________________________________________________________
Exercises and Projects — Creating a CD Jewel Case and Storage Box

You are part of a design team. The project manager has provided the following design criteria for a CD storage box:

- The CD storage box is constructed of a polymer (plastic) material.
- The storage box must hold 25 CD jewel cases.
- The title of the CD must be visible when the jewel case is positioned in the storage box.
- The wall thickness of the storage box is 1cm.
- On each side of the storage box, there must be 1cm clearance between the jewel case and the inside of the box.
- There must be 2cm clearance between the top of the CD cases and the inside of the storage box.
- There must be 2cm clearance between the jewel cases and the front of the storage box.

Task 1 — Measuring the CD Jewel Case

Measure the width, height, and depth of one CD jewel case. What are the measurements in centimeters?

**Answer:**

Width: _____________________________

Height: ____________________________

Depth: _____________________________

Task 2— Rough Sketch of the Jewel Case

Using paper and pencil, manually sketch the CD jewel case. Label the dimensions.

Task 3 — Calculate the Overall Case Capacity

Calculate the overall size of 25 stacked CD jewel cases. Record the overall width, height and depth.

**Answer:**

Overall width: __________________________

Overall height: _________________________

Overall depth: _________________________
Lesson 3: The 40-Minute Running Start

Task 4—Calculate the Outside Measurements of the CD Storage Box

Calculate the overall outside measurements of the CD storage box. The box requires a clearance to insert and position the CD jewel cases. Add a 2cm clearance to the overall width (1cm on each side) and 2cm to the height. The wall thickness is equal to 1cm.

Answer:

_______________________________________________________________________

Task 5—Creating the CD Jewel Case and Storage Box

Create two parts using SolidWorks.

☐ Model a CD jewel case. You should use the dimensions you obtained in Task 1. Name the part CD case.

Note: A real CD jewel case is an assembly of several parts. For this exercise, you will make a simplified representation of a jewel case. It will be a single part that represents the overall outside dimensions of the jewel case.

☐ Design a storage box to hold 25 CD jewel cases.

☐ Save both parts. You will use them to make an assembly at the end of the next lesson.

More to Explore — Modeling More Parts

Description

Look at the following examples. There are at least three features in each example. Identify the 2D Sketch tools used to create the shapes. You should:

☐ Consider how the part should be broken down into individual features.

☐ Focus on creating sketches that represent the desired shape. You do not need to use dimensions. Concentrate on the shape.

☐ Also, experiment and create your own designs.

Note: Each new sketch should overlap an existing feature.
Task 1

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.

Task 2

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.

Task 3
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
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.

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.

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.

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

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.
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.
5. Create a Coincident mate between the back face of the cdcase and the inside back face of the storagebox.
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.
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 \( \text{ Concentric } \). 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.
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.

Click Insert, Component Pattern.
The Pattern Type dialog is displayed.

Click Define your own pattern (Local).
Select the option Arrange in a circular fashion (Circular).

Click Next.
The Local Component Pattern dialog box is displayed.

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.

Set the Spacing to 120°.
Set the Instances to 3.

Click View, Temporary Axes.

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.

Click Finish.

Turn off the temporary axes.

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

Click Move Component. 
Drag the Collar up and down while observing the motion of the assembly.

Save and close the assembly.
Lesson 4 Vocabulary Worksheet

Fill in the blanks with the words that are defined by the clues.

1. _______ copies one or more curves into the active sketch by projecting them onto the sketch plane.

2. In an assembly, parts are referred to as: ___________________________________________________________________________

3. Relationships that align and fit components together in an assembly: ____________________________

4. The symbol (f) in the FeatureManager design tree indicates a component is: ____________

5. The symbol (-) indicates a component is: __________________________________________________________________________

6. When you make a component pattern, the component you are copying is called the ____________ component.

7. A SolidWorks document that contains two or more parts: ____________________________

8. You cannot move or rotate a fixed component unless you __________________________ it first.
Lesson Summary

- An assembly contains two or more parts.
- In an assembly, parts are referred to as *components*.
- Mates are relationships that align and fit components together in an assembly.
- Components and their assembly are directly related through file linking.
- Changes in the components affect the assembly and changes in the assembly affect the components.
- The first component placed into an assembly is fixed.
- Under defined components can be moved using dynamic assembly motion. This simulates the movement of mechanisms.
Lesson 5: Toolbox Basics

Goals of This Lesson

- Place standard Toolbox parts in assemblies.
- Modify Toolbox part definitions to customize standard Toolbox parts.

Before Beginning This Lesson

- Complete the previous lesson — Assembly Basics.
- Verify that Toolbox and Toolbox Browser are set up and running on your computer. Toolbox and Toolbox Browser are SolidWorks add-ins which are not loaded automatically. These add-ins must be specifically added during installation.

Resources for This Lesson

This lesson plan corresponds to the Toolbox module in the SolidWorks Online Tutorials. For more information about the Online Tutorials, See “Online Tutorials” on page v.
Active Learning Exercises — Adding Toolbox Parts

Follow the instructions in the Toolbox module in the SolidWorks Online Tutorials. Then proceed with the exercise below.

Add screws to the switchplate using the predefined hardware in Toolbox.

In the previous lesson, you added screws to the switchplate by modeling the screws and mating them to the switchplate in an assembly. As a general rule, hardware — such as screws — are standard components. Toolbox gives you the ability to apply standard hardware to assemblies without having to model it first.

Open the Switchplate Toolbox Assembly

Open the Switchplate Toolbox Assembly.

Notice that this assembly only has one part — or component — in it. Switchplate is the only part in the assembly.

An assembly is where you combine parts together. In this case, you are adding the screws to the switchplate.
Open Toolbox Browser

Click \( \text{Open Toolbox Browser} \). It is the fourth tab at the bottom of the document window.

The Toolbox Browser appears.
The Toolbox Browser is the window that contains all available Toolbox parts.
The Toolbox Browser is organized like a traditional paper catalog.

Selecting the Appropriate Hardware

Toolbox contains a wide variety of hardware. Selecting the right hardware is often critical to the success of a model.

You must determine the size of the holes before selecting the hardware to use and match the hardware to the hole.

1. Click \( \text{Catalog} \) or \( \text{Chapter} \) and select one of the holes on the switchplate to determine the hole size.

Note: The dimensions in this lesson are shown in inches.
2 In the Toolbox Browser, select **Ansi Inch, Bolts and Screws**, and **Machine Screws** from the lists.

The valid types of machine screws display.

3 Click and hold **Pan Head (Cross)**.

Does this hardware selection make sense for this assembly? The switchplate was designed with the size of the fasteners in mind. The holes in the switchplate are specifically designed for a standard fastener size.

The fastener size is not the only consideration in selecting a part. The type of fastener is important too. For example, you would not use miniature screws or square head bolts for the switchplate. They are the wrong size. They would be either too small or too large. You also have to take into consideration the user of this product. This switchplate has to be attachable with the most common of household tools.

**Placing Hardware**

1 Drag the screw towards the switchplate.

As you begin to drag the screw, it may appear very large.

**Note:** Drag and drop parts by holding the left mouse button. Release the mouse button when the part is correctly oriented.
2 Slowly drag the screw towards one of the switchplate holes until the screw snaps into the hole.
When the screw snaps into the hole, it is correctly oriented and properly mates with the surfaces of the part that it is combined with.
The screw still may appear too large for the hole.
3 When the screw is in the correct position, release the mouse button.

Specifying the Properties of the Toolbox Part

After you release the mouse button, pop-up window appears. This window allows you to edit the screw properties.
1 If necessary, change the properties of the screw to match the holes. In this case, a #6-32 screw works with these holes.
2 When you have completed the property changes, click **OK**.
The first screw is now placed in the first hole.

3 Repeat the process for the second hole.
You should not have to change any of the screw properties for the second screw. Toolbox remembers your last selection.
Both screws are now in the switchplate.
5 Minute Assessment – Answer Key

1. How would you determine the size of a screw to place in an assembly?

2. In which window do you find ready-to-use hardware components?

3. True or False: Parts from Toolbox automatically size to the components they are being placed on.

4. True or False: Toolbox parts can only be added to assemblies.

5. How can you resize components as you are placing them?
Exercises and Projects — Bearing Block Assembly

Add bolts and washers to fasten the bearing rest to the bearing block.

Opening the Assembly

1. Open Bearing Block Assembly.

Bearing Block Assembly has Bearing Rest and Bearing Block as components.

In this exercise, you are going to bolt the bearing rest to the bearing block. The through holes in the bearing rest are designed to allow the bolts to pass through but not be loose. The holes in the bearing block are tapped holes. Tapped holes are threaded and specifically designed to act like nuts do. In other words, the bolt screws directly into the bearing block.

If you take a close look at the holes, you see that the holes in the bearing rest are larger than those of the bearing block. That is because the holes in the bearing block are represented with the amount of material needed for the creation of the screw threads. The screw threads are not visible. Threads are rarely shown in models.

Placing Washers

Washers have to be placed before the screws or bolts. You do not have to use washers every time you place screws. However, when you do intend to use washers, they must be placed before screws, bolts, or nuts so that the correct relationships can be established.

The washers mate with the surface of the part and the screw or bolt mates with the washer. Nuts also mate with washers.

2. Click 💧.

The Toolbox Browser appears.
3 In the Toolbox Browser, select **Ansi Inch, Washers, and Plain Washers (Type A)** from the lists.

The valid types of Type A Washers display.

4 Click and hold **Preferred - Narrow** washer.

5 Slowly drag the washer towards one of the bearing rest through holes until the washer seems to snap onto the hole.

When the washer snaps onto the hole, it is correctly oriented and properly mates with the surfaces of the part that it is combined with.

The washer still may appear too large for the hole.

6 When the washer is in the correct position, release the mouse button.

After you release the mouse button, a pop-up window appears. This window allows you to edit the properties of the washer.

7 Edit the washer properties for a 3/8th hole and click **OK**.

The washer is placed.

Notice that the inside diameter is slightly larger than 3/8th. In general, the size of the washer indicates the size of the bolt or screw that must pass through it — not the actual size of the washer.

8 Place a washer on the other hole.
Placing Screws

1. Select **Ansi Inch, Bolts and Screws**, and **Machine Screws** from Toolbox Browser.

2. Drag a hex screw to one of the washers that you placed earlier.

3. Snap the screw into place and release the mouse button. A window appears with the properties for the hex screw.

4. Select a 3/8-24 screw of the appropriate length and click **OK**.

   The first screw is placed. The screw establishes a mate relationship with the washer.

5. Place the second screw in the same way.

### Thread Display

While fasteners such as bolts and screws are fairly detailed parts, they also very common ones. In general, bolts and screws are not the parts that you design. Instead you will use off-the-shelf hardware components. It is a well-established design practice to not draw all of the details of fasteners, but to specify their properties and show only an outline — or simplified — view of them.

The three display modes for bolts and screws are:

- **Simplified** — Represents the hardware with few details. Most common display. Simplified display shows the bolt or screw as if it were unthreaded.

- **Cosmetic** — Represents some details of the hardware. Cosmetic display shows the barrel of the bolt or screw and represents the size of the threads as dashed lines.

- **Schematic** — Very detailed display which is rarely used. Schematic shows the bolt or screw as it really appears. This display is best used when designing a unique fastener or when specifying an uncommon one.
Making Sure That the Screws Fit

Before you placed the washers and screws, you should have measured the depth of the holes and the thickness of the washer as well as the diameter of the holes.

Even if you measured before placing the hardware, it is a good practice to verify that the screw fits as you intended it to. Viewing the assembly in wireframe, viewing it from different angles, using Measure, or creating a section view are some ways to do this.

A section view lets you look at the assembly as if you took a saw and cut it open.

1. Click Section View.
   The Section View window appears.

2. Select the Right reference plane from the FeatureManager design tree.

3. Specify Right as the Section Plane.

4. Specify 3.4175 as the Section Position.

5. Click OK.

Now you see the cut away of the assembly right down the center of one of the screws. Is the screw long enough? Is it too long?

Modifying Toolbox Parts

If the screws — or other parts placed from Toolbox — are not the correct size you can modify their properties.

1. Select the part to modify, right-click, and select Edit Toolbox Definition.
   The Edit Toolbox Definition window appears. This window has appeared before. It is the window that you used to specify the properties of Toolbox parts as you were placing them.

2. Modify the part properties and click OK.
   The Toolbox part changes.

Note: After modifying parts, you should rebuild the assembly.
More to Explore

In the previous exercise you used Toolbox to add washers and screws to an assembly. In that assembly, the screws went into blind holes. In this exercise, add washers, lock washers, screws, and nuts to an assembly.

1. Open Bearing Plate Assembly.
2. Add the washers to the through holes on the bearing rest first. The holes are 3/8th diameter.
3. Add the lock washers to the far side of the plate next.
4. Add 1-inch machine screws. Snap these to the washers on the bearing rest.
5. Add hex nuts. Snap these to the lock washers.
6. Use the techniques that you have learned to verify that the hardware is the correct size for this assembly.
Lesson 5 Vocabulary Worksheet

Name: _______________________________ Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided.

1. View that lets you look at the assembly as if you took a saw and cut it open: __________
   _______________________________________________________________________

2. Type of hole that allows a screw or bolt to be screwed directly into it: ______________
   _______________________________________________________________________

3. Common design practice that represents the screws and bolts showing outlines and few details:
   _______________________________________________________________________

4. Method for moving a Toolbox part from the Toolbox Browser to the assembly: ______
   _______________________________________________________________________

5. Window that contains all available Toolbox parts: _____________________________

6. A file where you combine parts together: _________________________________

7. Hardware — such as screws, nuts, washers, and lock washers — that you can select from the Toolbox Browser: _____________________________

8. Type of hole that allows a screw or bolt into it, but is not tapped: ______________
   _______________________________________________________________________

9. Properties — such as size, length, thread length, display type — that describe a Toolbox part: _____________________________
Lesson Summary

- Toolbox provides ready-to-use parts — such as bolts and screws.
- Toolbox parts are placed by dragging and dropping them in assemblies.
- You can edit the property definitions of Toolbox parts.
- Holes created with the hole wizard are easy to match with properly-sized hardware from Toolbox.
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?
6 How do you move a drawing view?
_____________________________________________________________________
_____________________________________________________________________

7 What command is used to import part dimensions into the drawing?
_____________________________________________________________________
_____________________________________________________________________

8 True or False. Dimensions must be clearly positioned on the drawing.
_____________________________________________________________________

9 Give four rules for good dimensioning practice.
_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________
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 **Create** 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 $\text{A}$ 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 $\text{[ ]}$. 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.

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

```
WALL THICKNESS = 4mm
```
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, 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 7: eDrawing Basics

Goals of This Lesson

- Create eDrawings from existing SolidWorks files.
- View and manipulate eDrawings.
- Email eDrawings.

Before Beginning This Lesson

- Complete the previous lesson — Drawing Basics.
- An email application has to be loaded on your computer. If email is not present on your computer, you will not be able to complete More to Explore which is an exercise that teaches you how to email an eDrawing.
- Verify that eDrawings2003 is set up and running on your computer. eDrawings is a SolidWorks add-in which is not loaded automatically. This add-in must be specifically added during installation.

Resources for This Lesson

This lesson plan corresponds to the eDrawings module in the SolidWorks Online Tutorials. For more information about the Online Tutorials, See “Online Tutorials” on page v.

eDrawing Toolbars

By default, when the eDrawings viewer starts, the toolbars are displayed with large buttons like this . This makes it easier to learn what the buttons do. However, you might want to use smaller buttons like this to save screen space. To use small buttons, click View, Toolbars, Large Buttons in the eDrawings viewer. Clear the check mark in from of the menu listing.

The remaining illustrations in this lesson are shown with small buttons.
Active Learning Exercises

Follow the instructions in the eDrawings module in the SolidWorks Online Tutorials. Then proceed with the exercises below.

Create and explore an eDrawing of the switchplate part created earlier.

Creating an eDrawing

1. In SolidWorks, open the switchplate part.

   **Note:** You created switchplate during Lesson 2.

2. Click **Publish an eDrawing 2003** on the eDrawings toolbar to publish an eDrawing of the part.

   The eDrawing of switchplate appears in the eDrawings Viewer.

   **Note:** You can create eDrawings from AutoCAD® drawings too. Refer to the topic Creating eDrawing Files in the eDrawings online help for more information.

Using Quick Help

Context-sensitive Quick Help boxes provide on-screen help with tasks and tools. Once you use a tool described by Quick Help, the software assumes you have learned how to use that tool, and that Quick Help box disappears. Quick Help does not reappear if you use that same tool later.

To toggle Quick Help on or off, click **Help, Quick Help**. Quick Help is on when a check mark appears in front of the menu listing.

**Note:** If you turn Quick Help off and then turn it back on, all Quick Help boxes are reactivated.
Viewing an Animated eDrawing

Animation allows you to dynamically view eDrawings.

1. Click **Next**.
   The view changes to the Front view. You can click **Next** repeatedly to step through the views.

2. Click **Previous**.
   The previous view is displayed.

3. Click **Continuous Play**.
   Each view is displayed one by one in a continuous display.

4. Click **Stop**.
   The continuous display of views halts.

5. Click **Home**.
   The default or home view is displayed.

Viewing Shaded and Wireframe eDrawings

1. Click **Shaded**.
   The display of the switch plate changes from shaded to wireframe.

2. Click **Shaded** again.
   The display of the switch plate changes from wireframe to shaded.

Saving an eDrawing File

1. In the eDrawings Viewer click **File, Save As**.

2. Select **Enable measure**.
   This option allows anyone viewing the eDrawing file to measure the geometry. This is called making the file “review-enabled”.

3. Select **Compress file**.
   This option compresses the eDrawing file size by approximately 40%, but may decrease the quality of some shaded images. If you do not select this option, the shaded image quality matches the quality in SolidWorks. When you work with eDrawings, you have to decide which is more important: file size or image quality.

4. Click **Save**.
Measuring and Markup

You can markup eDrawings with tools from the Markup toolbar. For tracking purposes markup comments appear as discussion threads on the Markup tab of the eDrawing Manager. In this example you will add a cloud with text and a leader.

1. Click **Cloud with Leader** on the Markup toolbar.
   Move the cursor into the graphics area. The pointer changes to .

2. Click the front face of the **switchplate**.
   This is where the leader will begin. A text box appears.

3. In the text box, type the text you want to appear in the cloud and then click **OK**.

4. Move the pointer to where you want to place the text and then click.
   The cloud with text appears attached to the leader. If necessary, click **Zoom to Fit**.

5. Close the eDrawing file, saving your changes.
5 Minute Assessment – Answer Key

1. How do you create an eDrawing?

2. How do you send others eDrawings?

3. What is the quickest way to return to the default view?

4. True or False: You can make changes to a model in an eDrawing.

5. True or False: You need to have the SolidWorks application in order to view eDrawings.

6. What eDrawings feature allows you to dynamically view parts, drawings, and assemblies?
In this exercise, you explore eDrawings created from SolidWorks parts, assemblies, and drawings.

**eDrawings of Parts**

1. In SolidWorks, open the **Tutor1** part created in Lesson 3.
2. Click **Publish an eDrawing 2003**.
   An eDrawing of the part appears in the eDrawings Viewer.

3. Hold **Shift** and press one of the arrow keys.
   The view of rotates 90° each time you press an arrow key.
4. Press an arrow key without holding **Shift**.
   The view of rotates 15° each time you press an arrow key.
5. Click **Home**.
   The default or home view is displayed.
6. Click **Continuous Play**.
   Each view is displayed one by one in a continuous display. Observe this for a moment.
7. Click **Stop**.
   The continuous display of views halts.
8. Close the eDrawing file without saving it.
eDrawings of Assemblies

1. In SolidWorks, open the Tutor assembly created in Lesson 4.

2. Click Publish an eDrawing 2003.
   
   An eDrawing of the assembly appears in the eDrawings Viewer.

3. Click Continuous Play.
   
   Each view is displayed one by one. Observe this for a moment.

4. Click Stop.
   
   The continuous display of views halts.

5. Click Home.
   
   The default or home view is displayed.
6 In the **Components** panel, right-click **Tutor1-1** and select **Make Transparent** from the shortcut menu.

The **Tutor1-1** part become transparent so you can see through it.

7 Right-click **Tutor1-1** and select **Hide** from the shortcut menu.

The **Tutor1-1** part no longer displays in the eDrawing. This part still exists in the eDrawing, it is just hidden.

8 Right-click **Tutor1-1** again and select **Show**.

The **Tutor1-1** part displays.
eDrawings of Drawings

1. Open the drawing you created Lesson 6. This drawing has two sheets. Sheet 1 shows the part Tutor1. Sheet 2 shows the Tutor assembly. An example of this is in the Lesson06 folder and is named Finished Drawing.

2. Click **Publish an eDrawing 2003**.

3. Select **All sheets**.

   A window appears so you can select which sheets to include in the eDrawing.

   Click **OK**.

   An eDrawing of the drawing appears in the eDrawings Viewer.

4. Click **Continuous Play**.

   Each view is displayed one by one. Observe this for a moment. Notice that the animation stepped through both sheets of the drawing.

5. Click **Stop**.

   The continuous display of drawing views halts.

6. Click **Home**.

   The default or home view is displayed.
Using the eDrawing Manager

You can use the eDrawing Manager, located on the left side of the eDrawings Viewer, to display tabs that let you manage file information. When you open a file, the most appropriate tab is automatically active. For example, when you open a drawing file, the Sheets tab is active.

The Sheets tab makes it easy to navigate through a multi-sheet drawing.

1. In the Sheets tab of the eDrawing Manager, double-click Sheet2.

   Sheet2 of the drawing is displayed in the eDrawings Viewer. Use this method to navigate a multi-sheet drawing.

   **Note:** You can also switch between multiple sheets by clicking the tabs located below the graphics area.

2. In the Sheets tab of the eDrawing Manager, right-click Drawing View9.

   The Hide/Show menu appears.

3. Click Hide.

   Notice how the eDrawing changes.

4. Return to Sheet1.

The 3D Pointer

You can use the 3D Pointer to point to a location in all of the drawing views in drawing files. When you use the 3D Pointer, linked crosshairs appear in each of the drawing views. For example, you can place the crosshairs on an edge in one view and the crosshairs in the other views point to the same edge.
The crosshairs colors indicate the following:

<table>
<thead>
<tr>
<th>Color</th>
<th>Axis</th>
</tr>
</thead>
<tbody>
<tr>
<td>Red</td>
<td>X-Axis (perpendicular to YZ plane)</td>
</tr>
<tr>
<td>Blue</td>
<td>Y-Axis (perpendicular to XZ plane)</td>
</tr>
<tr>
<td>Green</td>
<td>Z-Axis (perpendicular to XY plane)</td>
</tr>
</tbody>
</table>

1. Click 3D Pointer.
   The eDrawing of the drawing displays the 3D pointer. The 3D pointer helps you to see the orientation of each view.

2. Move the 3D Pointer.
   Notice how the pointer moves in each view.

Overview Window

The Overview Window gives you a thumbnail view of the entire drawing sheet. This is especially handy when working with large, complicated drawings. You can use the navigate among the views. In the Overview Window, click the view you want to look at.

1. Click Overview Window.
   The Overview Window appears.

2. Click the Front view in the Overview Window.
   Notice how the eDrawings Viewer changes.
Lesson 7: eDrawing Basics

More to Explore

If your system is set up with an email application, you can see how easy it is to send an eDrawing to someone else.

1. Open one of the eDrawings that you created earlier in this lesson.
2. Click Send.
   
   The Send As menu appears.
3. Select the file type to send and click OK.
   
   An email message is created with the file attached.
4. Specify an email address to send the message to.
5. Add text to the email message if you would like to.
6. Click Send.
   
   The email is sent with the eDrawing attached. The person receiving it can view it, animate it, send it on to others, and so forth.
Lesson 7 Vocabulary Worksheet

Name: _______________________________ Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided.

1 The ability to dynamically view an eDrawing: ________________________________

2 Halting a continuous play of an eDrawing animation:___________________________

3 Command that allows you to step backwards one step at a time through an eDrawing animation:_____________________________________________________________

4 Non-stop replay of eDrawing animation: _____________________________________

5 Rendering of 3D parts with realistic colors and textures: ________________________

6 Go forward one step in an eDrawing animation: _______________________________

7 Command used to create an eDrawing: ______________________________________

8 Graphic aid that allows you to see the model orientation in an eDrawing created from a SolidWorks drawing: _____________________________________________________

9 Quickly return to the default view:__________________________________________

10 Command that allows you to use email eDrawings with others:_______________
Lesson Summary

- eDrawings can be created quickly from part, assembly, and drawing files.
- You can share eDrawings with others — even if they don’t have SolidWorks.
- Email is the easiest way to send an eDrawing to others.
- Animation allows you to see all views of a model.
- You can hide selected components of an assembly eDrawing and selected views of a drawing eDrawing.
Lesson 8: Design Tables

Goals of This Lesson

- Upon successful completion of this lesson, you will be able to create a design table that generates the following configurations of Tutor1:

![Design configurations](image)

<table>
<thead>
<tr>
<th></th>
<th>A</th>
<th>B</th>
<th>C</th>
<th>D</th>
<th>E</th>
<th>F</th>
<th>G</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Design Table for Tutor3</td>
<td>box_width@Sketch1</td>
<td>box_height@Sketch1</td>
<td>knob_dia@Sketch2</td>
<td>hole_dia@Sketch3</td>
<td>fillet_radius@Outside corners</td>
<td>Depth@Knob</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>blk1</td>
<td>120</td>
<td>120</td>
<td>70</td>
<td>50</td>
<td>10</td>
<td>50</td>
</tr>
<tr>
<td>4</td>
<td>blk2</td>
<td>120</td>
<td>90</td>
<td>50</td>
<td>40</td>
<td>15</td>
<td>30</td>
</tr>
<tr>
<td>5</td>
<td>blk3</td>
<td>90</td>
<td>150</td>
<td>60</td>
<td>10</td>
<td>30</td>
<td>15</td>
</tr>
<tr>
<td>6</td>
<td>blk4</td>
<td>120</td>
<td>120</td>
<td>30</td>
<td>10</td>
<td>25</td>
<td>90</td>
</tr>
</tbody>
</table>

Before Beginning This Lesson

- Design Tables requires Microsoft Excel application. Ensure that Microsoft Excel is loaded on your computer. It is strongly recommended that you use either Microsoft Office 2000, or Microsoft Excel 97 Service Release 2 (SR2) or later.
This lesson plan corresponds to the Design Tables module in the SolidWorks Online Tutorials. For more information about the Online Tutorials, See “Online Tutorials” on page v.
Active Learning Exercises — Creating a Design Table

Create the design table for Tuor1. Follow the instructions in the Design Tables module in the SolidWorks Online Tutorials.

<table>
<thead>
<tr>
<th></th>
<th>A</th>
<th>B</th>
<th>C</th>
<th>D</th>
<th>E</th>
<th>F</th>
<th>G</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Design Table for Tuor3</td>
<td>box_width@Sketch1</td>
<td>box_height@Sketch1</td>
<td>knob dia@Sketch2</td>
<td>hole dia@Sketch3</td>
<td>fillet_radius@Outside_corners</td>
<td>Depth@Knob</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>blk1</td>
<td>120</td>
<td>120</td>
<td>70</td>
<td>50</td>
<td>10</td>
<td>50</td>
</tr>
<tr>
<td>4</td>
<td>blk2</td>
<td>120</td>
<td>90</td>
<td>50</td>
<td>40</td>
<td>15</td>
<td>30</td>
</tr>
<tr>
<td>5</td>
<td>blk3</td>
<td>90</td>
<td>150</td>
<td>60</td>
<td>10</td>
<td>30</td>
<td>15</td>
</tr>
<tr>
<td>6</td>
<td>blk4</td>
<td>120</td>
<td>120</td>
<td>30</td>
<td>10</td>
<td>25</td>
<td>90</td>
</tr>
</tbody>
</table>

5 Minute Assessment – Answer Key

1. What is a configuration? ____________________________

2. What is a design table? ____________________________

3. What additional Microsoft software application is required to create design tables in SolidWorks? ____________________________

4. What are three key elements of a design table? ____________________________
5 True of False. **Link Values** equates a dimension value to a shared variable name.

6 Describe the advantage of using geometric relations versus linear dimensions to position the Knob feature on the Box feature. ________________________________________________
   ________________________________________________
   ________________________________________________
   ________________________________________________

7 What is the advantage of creating a design table? ________________________________________________
   ________________________________________________
   ________________________________________________
   ________________________________________________

**Exercises and Projects — Creating a Design Table for Tutor2**

**Task 1**

Create a design table for **Tutor2** that corresponds to the four configurations of **Tutor3**. Rename the features and the dimensions. Save the part as **Tutor4**.

**Task 2**

Create three configurations of the **CD storagebox** to contain 50, 100 and 200 CDs. The maximum width dimension is 120cm.

Some examples are sown at the right.
Task 3

Convert the overall dimensions of the 50 CD storagebox from centimeters to inches. The design for the CD storagebox was created overseas. The CD storagebox will be manufactured in the US.

Given:

- Conversion: $2.54\text{cm} = 1\text{ inch}$
- Box width = $54.0\text{cm}$
- Box height = $16.4\text{cm}$
- Box depth = $17.2\text{cm}$

Answer:

- Overall dimensions = $\text{box width} \times \text{box height} \times \text{box depth}$
- $\text{Box width} =$
- $\text{Box height} =$
- $\text{Box depth} =$

Task 4

What CD storagebox configurations are feasible for use in your classroom?
Lesson 8: Design Tables

Exercises and Projects — Creating Part Configurations Using Design Tables

Task 5

Create a cup. In the **Extrude Feature** dialog box, use a **5° Draft Angle**. Create four configurations using a design table. Experiment with different dimensions.

Task 6

Bring in some examples of products that lend themselves to design tables. You can bring in the actual objects or illustrations from magazines or catalogs.
More to Explore — Configurations, Assemblies, and Design Tables

When each component in an assembly has multiple configurations, it makes sense that the assembly should have multiple configurations as well. There are two ways to accomplish this:

- Manually change the configuration being used by each component in the assembly.
- Create an assembly design table that specifies which configuration of each component is to be used for each version of the assembly.

**Note:** If you followed the directions in the online tutorial, you saved Tutor1 as Tutor3 when you created the design table. Likewise in Task 1 of the exercises, Tutor2 would have been saved as Tutor4. To explore assembly design tables, you will need an assembly that is made up of Tutor3 and Tutor4.

**Changing the Configuration of a Component in an Assembly**

To manually change the displayed configuration of a component in an assembly:

1. Open the assembly Tutor Assembly which is located in the Lesson08 folder.
2. Right-click the component, either in the FeatureManager design tree or in the graphics area, and select **Component Properties**.
3. In the **Component Properties** dialog, select the desired configuration from the list in the **Referenced configuration** area. Click **OK**.
4. Repeat this procedure for each component in the assembly.
Assembly Design Tables

While manually changing the configuration of each component in an assembly works, it is neither efficient nor very flexible. Switching from one version of an assembly to another would be tedious. A better approach would be to create an assembly design table.

The procedure for creating an assembly design table is very similar to the procedure for creating a design table in an individual part. The most significant difference is the choice of different keywords for the column headers. The keyword we will explore here is $CONFIGURATION@component<instance>.

Procedure

1. Click Insert, Design Table. The Design Table PropertyManager appears.
2. For Source, click Blank and then click OK.
3. The Add Rows and Columns dialog box appears. If the assembly already contained configurations that were created manually they would be listed here. You could select them and they would automatically be added to the design table.
4. Click Cancel.
5. In cell B2, enter the keyword $Configuration@ followed by the name of the component and its instance number. In this example, the component is Tutor3 and the instance is <1>.
6. In cell C2, enter the keyword $Configuration@ Tutor4<1>.
7 Add the configuration names in column A.

8 Fill in the cells of columns B and C with the appropriate configurations for the two components.

9 Finish inserting the design table.
Click in the graphics area. The system reads the design table and generates the configurations.
Click OK to close the message dialog.

10 Switch to the ConfigurationManager.
Each of the configurations specified in the design table should be listed.

**Note:** The configuration names are listed in the ConfigurationManager alphabetically, not in the order in which they appeared in the design table.

11 Test the configurations.
Double-click on each configuration to verify that they display correctly.
Lesson Summary

- Design Tables simplify making families of parts.
- Design Tables automatically change the dimensions and features of an existing part to create multiple configurations. The configurations control the size and shape of a part.
- Design Tables requires Microsoft Excel application.
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? ______________________________________________________
   ____________________________________________________________
   ____________________________________________________________
   ____________________________________________________________
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:**

_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________
Lesson 9: Revolve and Sweep Features

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.

<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>15 inch candle</td>
</tr>
<tr>
<td>4</td>
<td>12 inch candle</td>
</tr>
<tr>
<td>5</td>
<td>10 inch candle</td>
</tr>
<tr>
<td>6</td>
<td>7 inch 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:

_______________________________________________________________________
_______________________________________________________________________
_______________________________________________________________________
_______________________________________________________________________
_______________________________________________________________________
_______________________________________________________________________
_______________________________________________________________________
_______________________________________________________________________

_______________________________________________________________________
_______________________________________________________________________
_______________________________________________________________________
_______________________________________________________________________
_______________________________________________________________________
_______________________________________________________________________
_______________________________________________________________________
Modify the outlet plate 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.
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.

- 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.
Lesson 10: Loft Features

Goals of This Lesson

☐ You will be able to create the following part:

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.

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.

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.

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.

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:

____________________________________________
____________________________________________
____________________________________________
____________________________________________
____________________________________________

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.
Lesson 11: Visualization

Goals of This Lesson

- You will create an image with PhotoWorks and an animation using SolidWorks Animator.

Before Beginning This Lesson

- This lesson requires copies of Tutor1, Tutor2 and the Tutor assembly. Tutor1, Tutor2 and the Tutor assembly were built earlier in the course.
- This lesson also requires the Claw-Mechanism that you built in Lesson 4.
- Verify that PhotoWorks release 2 and Animator are set up and running on your computer.

Resources for This Lesson

This lesson plan corresponds to the PhotoWorks and the SolidWorks Animator modules in the SolidWorks Online Tutorials. For more information about the Online Tutorials, See “Online Tutorials” on page v.
Active Learning Exercises — Using PhotoWorks

Follow the instructions in the *PhotoWorks* module in the Solidworks Online Tutorials. Then create a PhotoWorks rendering of *Tutor1* which you built in a previous lesson.

- Apply **Chrome** material.
- Set the **Background Style** to **Graduated**.
- Save the *Tutor Rendering.bmp* image.

The step-by-step instructions are as follows:

**Getting Started**

1. If **PhotoWorks** does not appear on the SolidWorks main menu bar, click **Tools, Add-Ins**, select **PhotoWorks release 2**, and click **OK**.

2. Click **Open** on the Standard toolbar, and open the part *Tutor1* which you built earlier.

3. Set the view orientation to **Isometric** and select **Shaded** view mode from the View toolbar. Your part should look like the illustration at the right.

**Shaded Rendering**

Shaded rendering is the basis for all photo-realistic rendering in PhotoWorks.

1. Click **Render** on the PhotoWorks toolbar.

The PhotoWorks software produces a smooth-shaded rendering of the part using a default material and scene.
Applying a Material

1. Click Material on the PhotoWorks toolbar. The Material Editor opens. The left pane is the Material Library where materials are listed in folders. The material tree shows all the folders currently loaded. Each folder can be expanded by clicking the plus sign next to it to show the sub-folders. The right panel is the Material Selection area.

2. Open the metals folder and then open the chrome sub-folder.

The material selection area displays a rendered image of a sphere for each material in the class.

3. Use the scroll bar to locate the chromium plate material.

4. Select the chromium plate material.

The Preview window is updated to display how the part will appear when it is rendered.

5. Click Apply.

6. Click Close.

7. Click Render.

The part is rendered with a chrome surface.

Set the Background Style to Graduated.

1. Click Scene on the PhotoWorks toolbar. The Scene Editor opens.

2. Open the backgrounds folder.

3. Open the graduated sub-folder.

4. Select graduated blue to white.

The Preview window updates.

5. Click Apply and Close.

6. Click Render.
What Makes an Image Look Realistic?

Highly reflective surfaces such as chrome are visually more interesting when there are details in the environment for them to reflect. Compare the image with the graduated background with the one that has the default background. Notice the reflections in the part.

Saving the Image

You can save a PhotoWorks image to a file for design proposals, technical documentation and product presentations. Images can be rendered to the following file types:

- Windows Bitmap (*.bmp)
- TIFF (*.tif)
- TARGA (*.tga)
- Mental Ray Scene file (*.mi)
- JPEG (*.jpg)
- PostScript (*.ps)
- Encapsulated PostScript (*.eps)
- Silicon Graphics 8-bit RGBA (*.rgb)
- Portable pixmap (*.ppm)
- Utah/Wavefront color, type A (*.rla)
- Utah/Wavefront color, type B (*.rlb)
- Softimage color (*.pic)
- Alias color (*.alias)
- Abekas/Quantel, PAL (720x576) (*.qntpal)
- Abekas/Quantel, NTSC (720x486) (*.qntntsc)
- Mental images, 8-bit color (*.ct)
To Save the Image:

1. Click **Render to File**.
2. In the **Render to File** window, specify a filename for the image.
3. In the **Format** field, specify a file type to save the image as.
4. Save the file in the directory as instructed by your teacher.
5. Optionally, you may set the **Width**, and **Height**.

**Note:** If you change the **Image size**, you should click **Fixed aspect ratio** to prevent distorting the image.

6. Click **Save**.
Active Learning Exercises – Creating an Animation

Create an animation of the lawn sprinkler. Follow the instructions in the *Solidworks Animator* module in the Solidworks Online Tutorials.

5 Minute Assessment – Answer Key

1. What is PhotoWorks?

2. List the rendering effects that are used in PhotoWorks?

3. The PhotoWorks __________________________ allows you to specify and preview materials.

4. Where do you set the scene background?

5. What is SolidWorks Animator?

6. List the three types of animations that can be created using the AnimationWizard.
Exercises and Projects — Creating an Exploded View of an Assembly

Using PhotoWorks and Animator Together

When you record an animation, the default rendering engine that is used is the SolidWorks shaded image software. This means the shaded images that make up the animation will look just like the shaded images you see in SolidWorks.

Earlier in this lesson you learned how to make photo-realistic images using the PhotoWorks application. You can record animations that are rendered using the PhotoWorks software. Since PhotoWorks rendering is much slower than SolidWorks shading, recording an animation this way takes much more time.

To use the PhotoWorks rendering software select PhotoWorks buffer from the Renderer: list on the Save Animation to File dialog box.

Note: The file types *.bmp and *.avi increase in file size as more materials and advanced rendering effects are applied. The larger the image size the more time is required to create the image and animation files.

Creating an Exploded View of an Assembly

The Claw-Mechanism which you used earlier already had an exploded view. To add an exploded view to an assembly, the Tutor assembly for example, follow this procedure:

1. Click Open on the Standard toolbar, and open the assembly, Tutor, which you built earlier.

2. Click Insert, Exploded View... or click Exploded View on the Assembly toolbar. The Assembly Exploder dialog box appears.
3 The **Step Editing** toolbar is used to create, edit, navigate through, delete, and apply explode steps. Each movement of a component in a single direction is considered a step.

4 Click **New** on the Step Editing toolbar to begin a new explode step. The dialog box expands to show selection lists for:
   - **Direction to explode along**
   - **Components to explode**
   - **Distance**

5 Click the flat face on the front of the **Tutor1** component. An arrow appears that is perpendicular to the selected face and the name **Face of Tutor1<1>** appears in the **Direction to explode along** list.

6 Select the **Tutor1** component, either by clicking it in the FeatureManager design tree, or the graphics area. The component name appears in the **Components to Explode** list.

7 Set the **Distance** to **70mm** and click **Apply** on the Step Editing toolbar.

8 Since there is only one component to explode, this completes making the exploded view.

9 Click **OK** to close the **Assembly Exploder** dialog box.
10 Results.

**Note:** Exploded views are related to and stored in configurations. You can only have one exploded view per configuration.

11 To collapse an exploded view, right-click in the FeatureManager design tree, and select **Collapse** from the shortcut menu.

12 To explode an existing exploded view, right-click the assembly icon in the FeatureManager design tree, and select **Explode** from the shortcut menu.
Exercises and Projects — Creating and Modifying Renderings

Task 1
Create a PhotoWorks rendering of Tutor2. Use the following settings:
- Use English Brick 2 material. Click the Texture tab and set the Scale to 0.5.
- Set the background to Plain White.
- Render and save the image.

Task 2
Modify the PhotoWorks rendering of Tutor1 that you created in the preceding Active Learning Exercise. Use the following settings:
- Change the material to Concrete from the Stones class.
- Set the background to Plain White.
- Render and save the image.

Task 3
Create a PhotoWorks rendering of the Tutor assembly. Use the following settings:
- Set the background to the scaled image Clouds.
- Render and save the image.

Task 4
Create PhotoWorks renderings of any of the parts and assemblies you built during class. For example, you might render the candlestick or the sports bottle you made created earlier. Experiment with different materials and scenes. You can try to create as realistic an image as possible, or you can create some unusual visual effects. Use your imagination. Be creative. Have fun.
Exercises and Projects — Creating an Animation

Create an animation that shows how the slides move relative to each other. In other words, create an animation where at least one of the slides moves. You cannot accomplish this task with the Animation Wizard.

1. Open the Nested Slides assembly. It is located in the Lesson11 folder.

2. Select the AnimationManager.

3. Select one of the slides. Select either Slide2 or Slide1. Slide3 is the first component of the assembly and is therefore fixed.

4. Click Create Path.
   The Create Path window appears.

5. Click Add Path Point.
   This establishes the starting point.

6. Click Move Component.

7. Move the slide and click Add Path Point again.

8. Create several more points in the same way.

9. Click Repeat initial path point as final path point.
   This causes the part to return to its starting position automatically.

10. When you have finished selecting points, click Done.
    The animation is now saved and ready to play back.

Exercises and Projects — Creating an Animation of the Claw-Mechanism

Create an animation of the Claw-Mechanism. Some suggestions include exploding and collapsing, and moving the Collar up and down to show assembly motion.

A completed copy of the Claw-Mechanism is located in the Lesson11 folder. This version is slightly different than the one you built in Lesson 4. This one does not have a component pattern. Each component was assembled individually. This is so the assembly will explode better.
More to Explore — Creating an Animation of Your Own Assembly

Earlier you created an animation from an existing assembly. Now create an animation using the Tutor assembly that you built earlier. The animation should include the following:

- Explode the assembly for a duration of 10 seconds.
- Rotate the assembly around the Y axis for a duration of 10 seconds.
- Collapse the assembly for a duration of 10 seconds.
- Record the animation. **Optional:** Record the animation using the PhotoWorks renderer.
Lesson Summary

- PhotoWorks and SolidWorks Animator create realistic representations of models.
- PhotoWorks uses realistic textures, materials, lighting, and other effects to produce true to life models.
- SolidWorks Animator animates and captures motion of SolidWorks parts and assemblies.
**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.
<table>
<thead>
<tr>
<th><strong>closed profile</strong></th>
<th>A closed profile (or closed contour) is a sketch or sketch entity with no exposed endpoints; for example, a circle or polygon.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>collapse</strong></td>
<td>Collapse is the opposite of explode. The collapse action returns an exploded assembly's parts to their normal positions.</td>
</tr>
<tr>
<td><strong>component</strong></td>
<td>A component is any part or sub-assembly within an assembly.</td>
</tr>
<tr>
<td><strong>configuration</strong></td>
<td>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.</td>
</tr>
<tr>
<td><strong>Configuration Manager</strong></td>
<td>The ConfigurationManager on the left side of the SolidWorks window is a means to create, select, and view the configurations of parts and assemblies.</td>
</tr>
<tr>
<td><strong>coordinate system</strong></td>
<td>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.</td>
</tr>
<tr>
<td><strong>degrees of freedom</strong></td>
<td>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.</td>
</tr>
<tr>
<td><strong>design table</strong></td>
<td>A design table is an Excel spreadsheet that is used to create multiple configurations in a part or assembly document. See configurations.</td>
</tr>
<tr>
<td><strong>document</strong></td>
<td>A SolidWorks document is a file containing a part, assembly, or drawing.</td>
</tr>
<tr>
<td><strong>drawing</strong></td>
<td>A drawing is a 2D representation of a 3D part or assembly. The extension for a SolidWorks drawing file name is .SLDDRW.</td>
</tr>
<tr>
<td><strong>drawing sheet</strong></td>
<td>A drawing sheet is a page in a drawing document.</td>
</tr>
<tr>
<td><strong>eDrawing</strong></td>
<td>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.</td>
</tr>
<tr>
<td><strong>face</strong></td>
<td>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.</td>
</tr>
<tr>
<td><strong>feature</strong></td>
<td>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.</td>
</tr>
<tr>
<td><strong>FeatureManager design tree</strong></td>
<td>The FeatureManager design tree on the left side of the SolidWorks window provides an outline view of the active part, assembly, or drawing.</td>
</tr>
<tr>
<td><strong>fillet</strong></td>
<td>A fillet is an internal rounding of a corner or edge in a sketch, or an edge on a surface or solid.</td>
</tr>
<tr>
<td><strong>graphics area</strong></td>
<td>The graphics area is the area in the SolidWorks window where the part, assembly, or drawing appears.</td>
</tr>
<tr>
<td><strong>helix</strong></td>
<td>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.</td>
</tr>
<tr>
<td><strong>instance</strong></td>
<td>An instance is an item in a pattern or a component that occurs more than once in an assembly.</td>
</tr>
<tr>
<td><strong>layer</strong></td>
<td>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.</td>
</tr>
<tr>
<td><strong>line</strong></td>
<td>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.</td>
</tr>
<tr>
<td><strong>loft</strong></td>
<td>A loft is a base, boss, cut, or surface feature created by transitions between profiles.</td>
</tr>
<tr>
<td><strong>mate</strong></td>
<td>A mate is a geometric relationship, such as coincident, perpendicular, tangent, and so on, between parts in an assembly. See also SmartMates.</td>
</tr>
<tr>
<td><strong>mategroup</strong></td>
<td>A mategroup is a collection of mates that are solved together. The order in which the mates appear within the mategroup does not matter.</td>
</tr>
<tr>
<td><strong>mirror</strong></td>
<td>(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.</td>
</tr>
<tr>
<td>Glossary Term</td>
<td>Definition</td>
</tr>
<tr>
<td>---------------</td>
<td>------------</td>
</tr>
<tr>
<td><strong>model</strong></td>
<td>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.</td>
</tr>
<tr>
<td><strong>mold</strong></td>
<td>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.</td>
</tr>
<tr>
<td><strong>named view</strong></td>
<td>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.</td>
</tr>
<tr>
<td><strong>open profile</strong></td>
<td>An open profile (or open contour) is a sketch or sketch entity with endpoints exposed. For example, a U-shaped profile is open.</td>
</tr>
<tr>
<td><strong>origin</strong></td>
<td>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.</td>
</tr>
<tr>
<td><strong>over defined</strong></td>
<td>A sketch is over defined when dimensions or relations are either in conflict or redundant.</td>
</tr>
<tr>
<td><strong>parameter</strong></td>
<td>A parameter is a value used to define a sketch or feature (often a dimension).</td>
</tr>
<tr>
<td><strong>part</strong></td>
<td>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.</td>
</tr>
<tr>
<td><strong>pattern</strong></td>
<td>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.</td>
</tr>
<tr>
<td><strong>planar</strong></td>
<td>An entity is planar if it can lie on one plane. For example, a circle is planar, but a helix is not.</td>
</tr>
<tr>
<td><strong>plane</strong></td>
<td>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.</td>
</tr>
<tr>
<td><strong>point</strong></td>
<td>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.</td>
</tr>
<tr>
<td><strong>profile</strong></td>
<td>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).</td>
</tr>
<tr>
<td><strong>Property Manager</strong></td>
<td>The PropertyManager is on the left side of the SolidWorks window for dynamic editing of sketch entities and most features.</td>
</tr>
<tr>
<td><strong>rebuild</strong></td>
<td>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.</td>
</tr>
<tr>
<td><strong>relation</strong></td>
<td>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.</td>
</tr>
<tr>
<td><strong>revolve</strong></td>
<td>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.</td>
</tr>
<tr>
<td><strong>section</strong></td>
<td>A section is another term for profile in sweeps.</td>
</tr>
<tr>
<td><strong>section view</strong></td>
<td>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.</td>
</tr>
<tr>
<td><strong>shaded</strong></td>
<td>A shaded view displays a model as a colored solid. See also HLR, HLG, and wireframe.</td>
</tr>
<tr>
<td><strong>sheet format</strong></td>
<td>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.</td>
</tr>
<tr>
<td><strong>shell</strong></td>
<td>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.</td>
</tr>
<tr>
<td><strong>sketch</strong></td>
<td>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.</td>
</tr>
<tr>
<td><strong>SmartMates</strong></td>
<td>A SmartMate is an assembly mating relation that is created automatically. See mate.</td>
</tr>
<tr>
<td><strong>sub-assembly</strong></td>
<td>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.</td>
</tr>
</tbody>
</table>
### Glossary

<table>
<thead>
<tr>
<th>Term</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>surface</strong></td>
<td>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.</td>
</tr>
<tr>
<td><strong>sweep</strong></td>
<td>A sweep creates a base, boss, cut, or surface feature by moving a profile (section) along a path.</td>
</tr>
<tr>
<td><strong>template</strong></td>
<td>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.</td>
</tr>
<tr>
<td><strong>toolbox</strong></td>
<td>A library of standard parts that are fully integrated with SolidWorks. These parts are ready-to-use components — such as bolts and screws.</td>
</tr>
<tr>
<td><strong>under defined</strong></td>
<td>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.</td>
</tr>
<tr>
<td><strong>vertex</strong></td>
<td>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.</td>
</tr>
<tr>
<td><strong>wireframe</strong></td>
<td>Wireframe is a view mode in which all edges of the part or assembly are displayed. See also HLR, HLG, shaded.</td>
</tr>
</tbody>
</table>