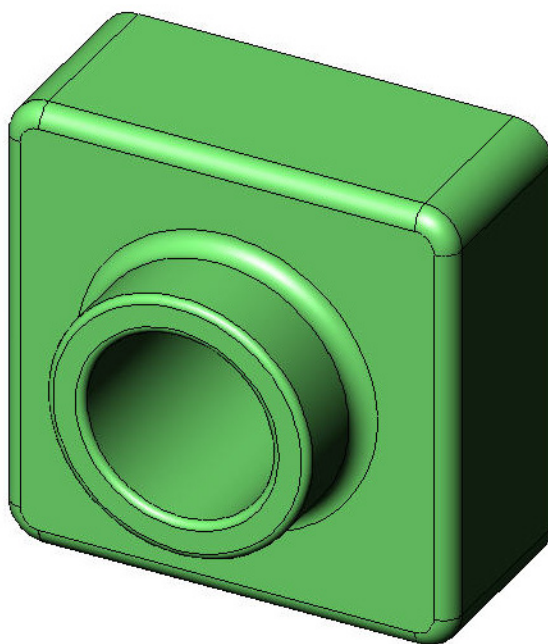


CAD Student Guide



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Introduction

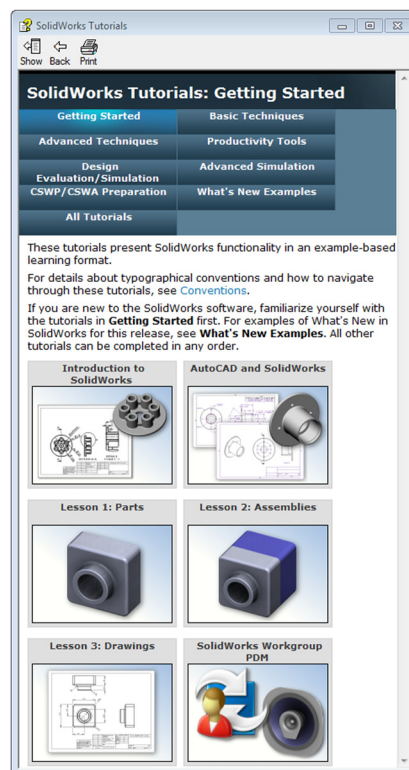
SolidWorks Tutorials

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

Accessing the SolidWorks Tutorials

To start the SolidWorks Tutorials, click **Help, SolidWorks Tutorials**. The SolidWorks window is resized and a second window appears next to it with a list of the available tutorials. There are over 40 lessons in the SolidWorks 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.

TIP: When you use SolidWorks Simulation to perform static engineering analysis, click **Help, SolidWorks Simulation, Tutorials** to access over 50 lessons and over 80 verification problems. Click **Tools, Add-ins** to activate SolidWorks Simulation.



Conventions

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

The following icons appear in the tutorials:

NEXT TOPIC: Click the link below “Next Topic:” to move to the next screen in the tutorial.



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



You can click most buttons that appear in the lessons to flash the corresponding SolidWorks button.



Open File automatically opens the file.



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.



Video: ... demonstrates with a video.

Printing the SolidWorks Tutorials

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

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

This displays the table of contents for the SolidWorks 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.

Focused Product Development Applications

There are additional SolidWorks Education packages that can be purchased that offer students more focused product development applications. These include:

☐ SolidWorks Electrical - Schematic

- Develop system-level electrical schematics needed to power and control your robot or student project. Use this powerful schematic tool to plan power, control, safety, PLC, and other systems for your design. (Note: Not for PCB assembly design.)

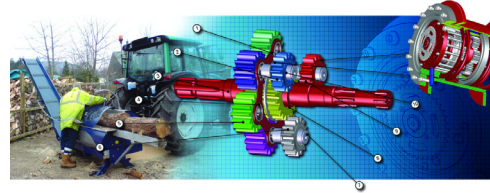


❑ SolidWorks Electrical - Professional

- Combines both electrical systems schematic design with 3D cable/wire/harness modeling capability in a single product to enable users to complete their overall product design. Educators and students can now design projects, including robots and Formula SAE cars, and include the necessary electrical aspects in their designs.

❑ SolidWorks Composer

- Use SolidWorks Composer to develop technical documentation that is linked to your original SolidWorks CAD files, streamlining development and management of product instructions, user manuals, and training materials for your product - just as professionals do in industry. Great for team projects including robot design, Formula SAE, and other educational projects.



❑ SolidWorks Enterprise PDM

- SolidWorks Enterprise PDM (EPDM) is a professional-grade data management tool available to schools for managing student design data and to introduce students to the world of product data management. Users can manage revisions, plan workflows and signoffs, and ensure documents are controlled. Manage your next team design project using SolidWorks Enterprise PDM.



Lesson 1: Using the Interface

Goals of This Lesson

- ☐ Become familiar with the Microsoft Windows® interface.
- ☐ Become familiar with the SolidWorks user interface.

Before Beginning This Lesson

- ☐ Verify that Microsoft Windows is loaded and running on your classroom/lab computers.
- ☐ Verify that the SolidWorks software is loaded and running on your classroom/lab computers in accordance with your SolidWorks license.
- ☐ Load the lesson files from the Educator Resources link.

Competencies for Lesson 1

You develop the following competencies in this lesson:

- ☐ **Engineering:** Knowledge of an engineering design industry software application.
- ☐ **Technology:** Understand file management, copy, save, starting and exiting programs.




SolidWorks education suite contains more than 80 eLearning tutorials in engineering design, simulation, sustainability, and analysis.

Active Learning Exercise — Using the Interface

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

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 **All Programs, SolidWorks, SolidWorks**.
The SolidWorks application program is now running.

TIP: A desktop shortcut is an icon that you can double-click to go directly to the file or folder represented. The illustration shows the SolidWorks shortcut.



Exit the Program

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


Opening an Existing File

- 3 Double-click on the SolidWorks part file Dumbell1 in the Lesson01 folder.
This opens the Dumbell1 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 **File, Open**, and typing or browsing to a file name or by selecting a file name from the **File** menu in SolidWorks. SolidWorks lists the last several files that you had open.

Saving a File

- 4 Click **Save**  on the Menu Bar 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

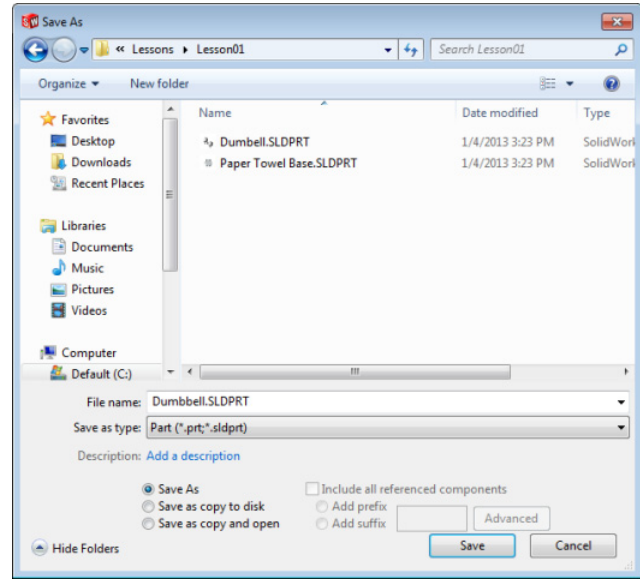
Notice that Dumbell is not spelled correctly. It is supposed to have two “b’s”.

- 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 change the name to 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

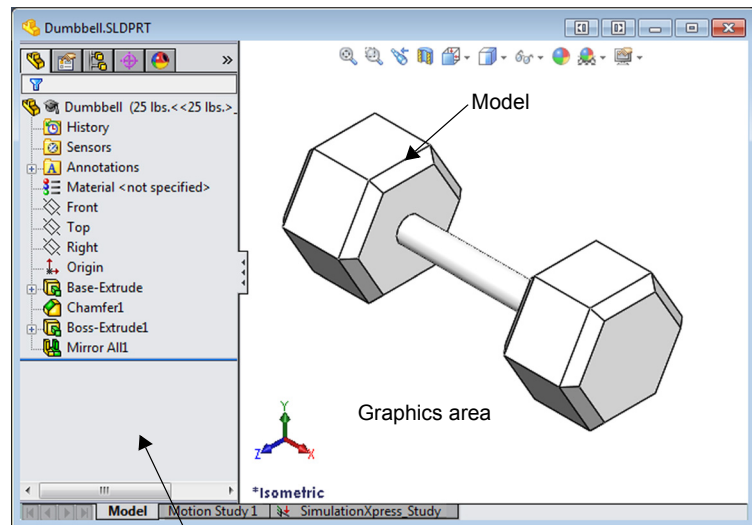
SolidWorks windows have two panels. One panel provides non-graphic data. The other panel provides graphic representation of the part, assembly, or drawing.

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

- 1 Click each of the tabs at the top 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 and in an isometric view. These are some of the ways in which the model can be represented very realistically.



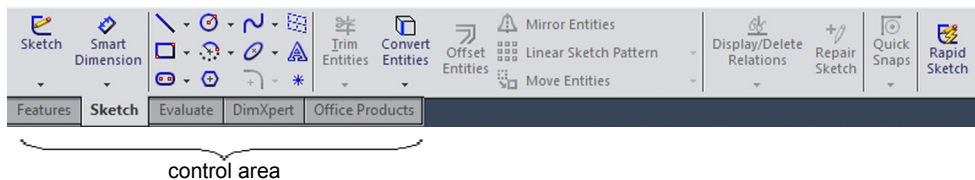
Left panel displaying the FeatureManager design tree

CommandManager

The CommandManager is a context-sensitive toolbar that dynamically updates based on the functions you want to access. By default, it displays tabs that are based on the document type. Use the CommandManager to access functions in a central location and to save space for the graphics area.

When you click a tab in the control area, the CommandManager updates to show those tools. For example, if you click **Sketch** in the control area, the sketch tools appear in the CommandManager. The convention for using the CommandManager is to write, “Click **Sketch** > **Smart Dimension** .

In this convention, **Sketch** is the CommandManager tab and **Smart Dimension** is the tooltip.




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 CommandManager.

Getting Online Help

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

- ☐ Click the flyout menu of Help options  in the menu bar.
- ☐ Click **Help, SolidWorks Help**.
- ☐ While in a command, click **Help**  in the dialog.

Lesson 1 — 5 Minute Assessment

Name: _____ Class: _____ Date: _____

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

1 How do you open the file from Windows Explorer?

2 How do you start the SolidWorks program?

3 What is the quickest way to start the SolidWorks program?

4 How do you copy a part within the SolidWorks program?

Lesson 1 Vocabulary Worksheet

Name: _____ Class: _____ Date: _____

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

- 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 Area of the screen that displays the work of a program: _____
- 6 Icon that you can double-click to start a program: _____
- 7 Action that quickly displays shortcut menus of frequently used or detailed commands:

- 8 Command that updates your file with changes that you have made to it: _____

- 9 Action that quickly opens a part or program: _____
- 10 The program that helps you create parts, assemblies, and drawings: _____
- 11 Panel of the SolidWorks window that displays a visual representation of your parts, assemblies, and drawings: _____

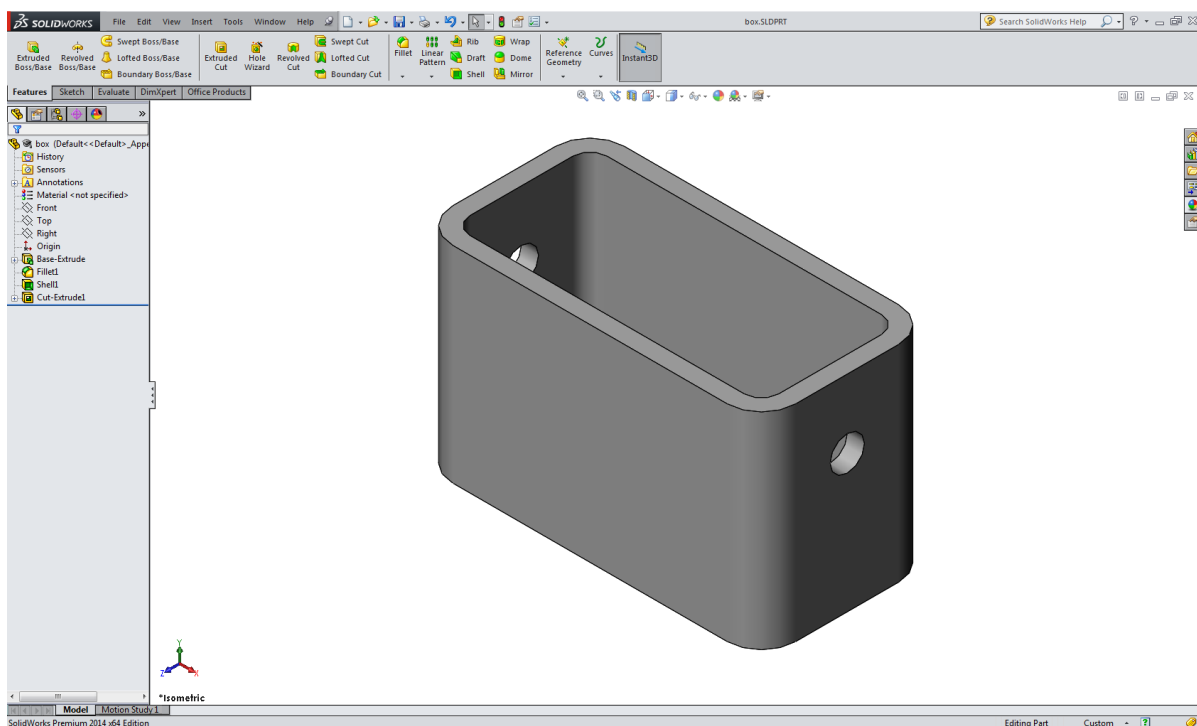
Lesson Summary

- ❑ The Start menu is where you go to start programs or find 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

- ❑ Understand the basic functionality of the SolidWorks software.
- ❑ Create the following part:



Before Beginning This Lesson

Complete Lesson 1: Using the Interface.



SolidWorks supports student teams in Formula Student, FSAE, and other regional and national competitions. For software sponsorship, go to www.solidworks.com/students

Competencies for Lesson 2

You develop the following competencies in this lesson:

- ❑ **Engineering:** Develop a 3D part based on a selected plane, dimensions, and features. Apply the design process to develop the box or switch plate out of cardboard or other material. Develop manual sketching techniques by drawing the switch plate.
- ❑ **Technology:** Apply a windows based graphical user interface.
- ❑ **Math:** Understand units of measurement, adding and subtracting material, perpendicularity, and the x-y-z coordinate system.


Active Learning Exercises — Creating a Basic Part

Use SolidWorks to create the box shown at the right.

The step-by-step instructions are given below.



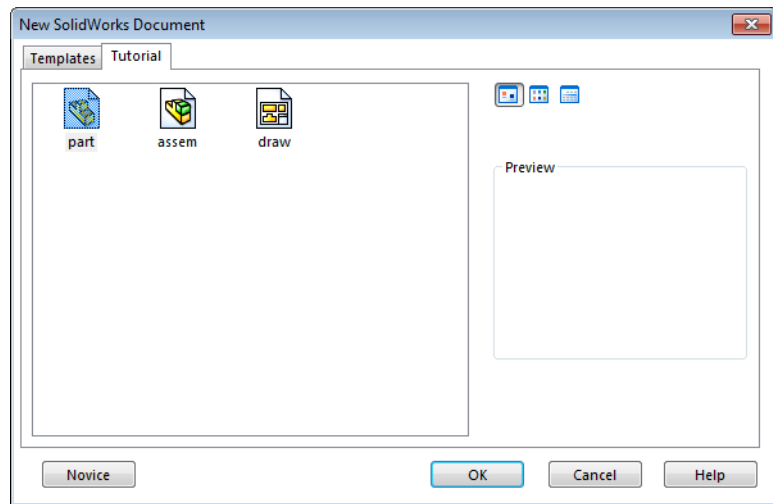
Create a New Part Document

- 1 Create a new part. Click **New**  on the Menu Bar.

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

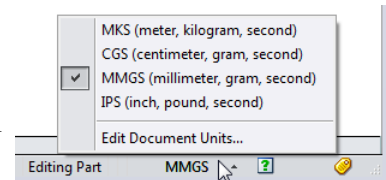
- 2 Click the **Tutorial** tab.
- 3 Select the **Part** icon.
- 4 Click **OK**.

A new part document window appears.



Setting Units

The units for a document can be set by clicking **Tools, Options** and then under the **Document Properties** tab, selecting units from the menu on the left. In the status bar, units can also be changed by clicking **Unit System** and then selecting the new unit system.




Base Feature

The Base feature requires:

- ☐ Sketch plane – Front
- ☐ Sketch profile – 2D Rectangle
- ☐ Feature type – Extruded boss feature

Open a Sketch

- 1 Click to select the **Front** plane in the FeatureManager design tree.
- 2 Open a 2D sketch. Click **Sketch > Sketch** .

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**.

Lesson 2: Basic Functionality

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 this symbol exits the sketch saving your changes. Clicking the red X exits the sketch discarding your changes.

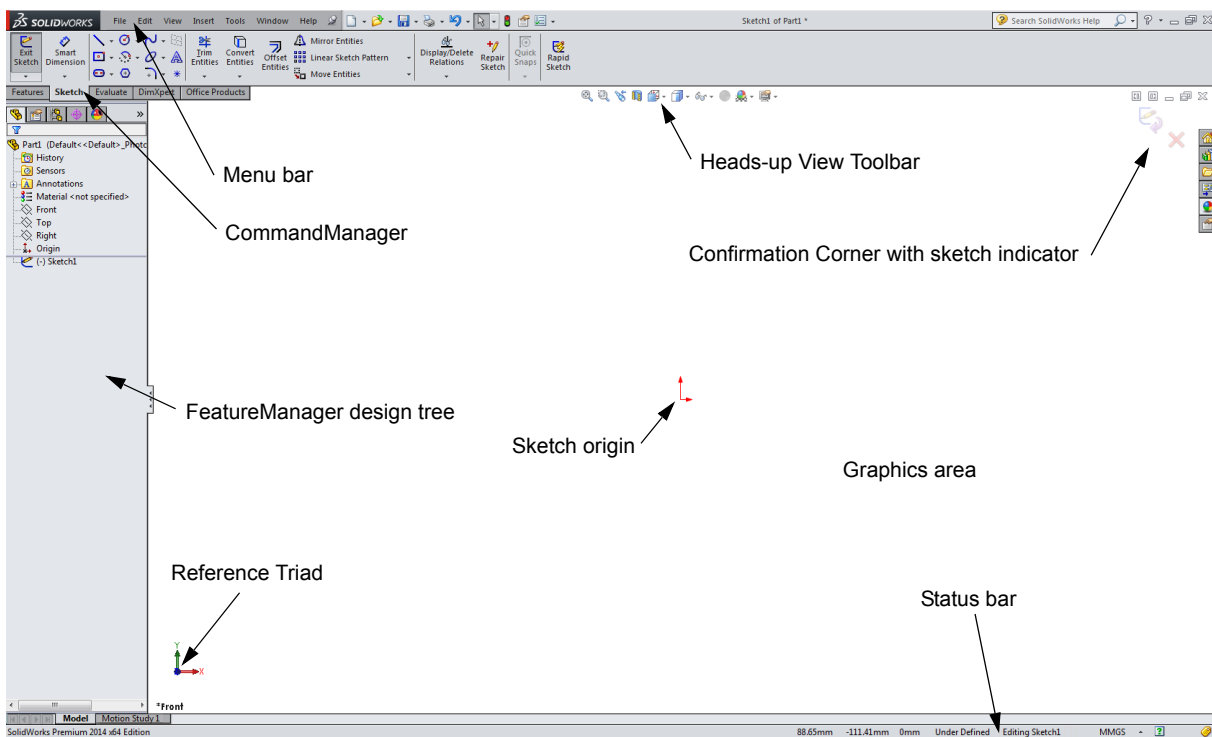


When other commands are active, the confirmation corner displays two symbols: a check mark and an X. The check mark executes the current command. The X cancels the command.




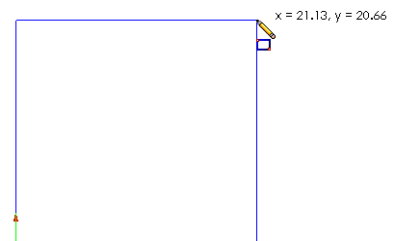
Overview of the SolidWorks Window

- ❑ A sketch origin appears in the center of the graphics area.
- ❑ **Editing Sketch1** 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.







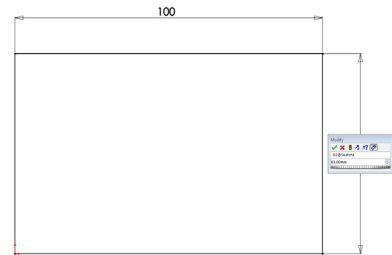
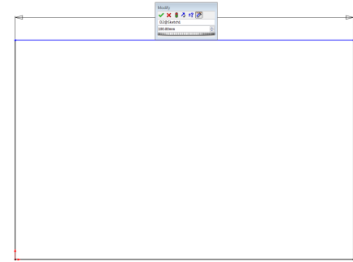
Sketch a Rectangle

- 1 Click **Sketch > Corner Rectangle** .
- 2 Click the sketch origin to start the rectangle.
- 3 Move the pointer up and to the right, to create a rectangle.
- 4 Click the mouse button again to complete the rectangle.




Add Dimensions

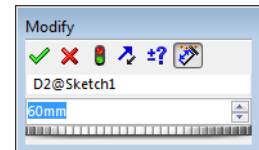
- 1 Click **Sketch > Smart Dimension** .
The pointer shape changes to .
 - 2 Click the top line of the rectangle.
 - 3 Click the dimension text location above the top line.
The **Modify** dialog box is displayed.
 - 4 Enter **100**. Click  or press **Enter**.
 - 5 Click the right edge of the rectangle.
 - 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.

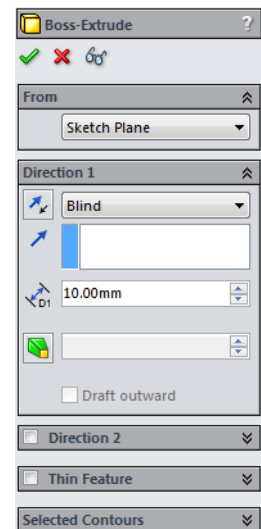
- 1 Double-click **65**.
The **Modify** dialog box appears.
- 2 Enter **60** in the **Modify** dialog box.
- 3 Click .



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.

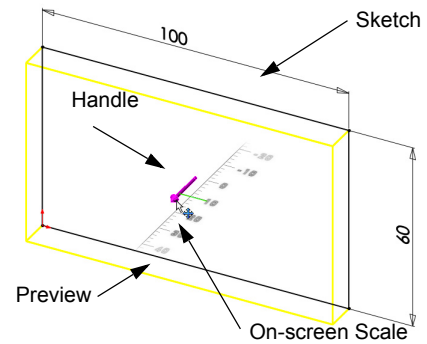
- 1 Click **Features > Extruded Boss/Base** .
The **Boss-Extrude** PropertyManager appears. The view of the sketch changes to trimetric.




2 Preview graphics.

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


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

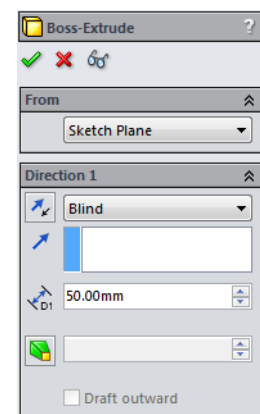


The cursor changes to . If you want to create the feature now, click the right mouse button. Otherwise, you can make additional changes to the settings. For example, the depth of extrusion can be changed by dragging the dynamic handle with the mouse or by setting a value in the PropertyManager.

3 Extrude feature settings.

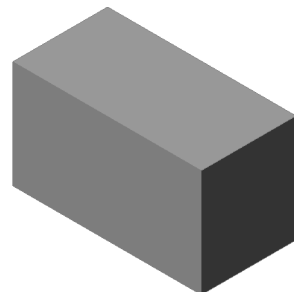
Change the settings as shown.

- End Condition = **Blind**
-  (Depth) = **50**



4 Create the extrusion. Click **OK** .

The new feature, Boss-Extrude1, is displayed in the FeatureManager design tree.



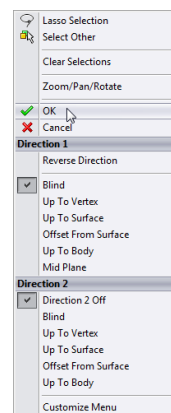
TIP:


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

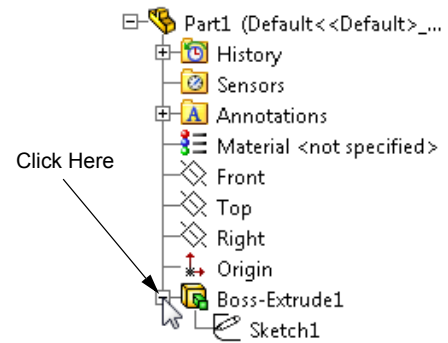
A second method is the set of **OK/Cancel** buttons in the confirmation corner of the graphics area.




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



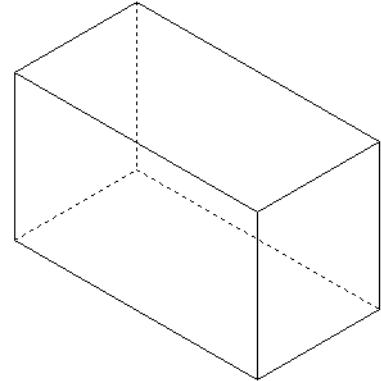
- Click the plus sign  beside Boss-Extrude1 in the FeatureManager design tree. Notice that Sketch1 — which you used to extrude the feature — is now listed under the feature.




View Display

Change the display mode. Click **Display Style > Hidden Lines Visible**  on the Heads-up View toolbar.

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




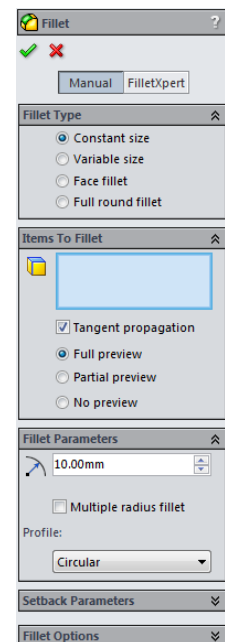
Save the Part

- Click **Save**  on the Menu Bar, or click **File, Save**.
The **Save As** dialog box appears.
- 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 (10 mm). Create them as a single feature.

- Click **Features > Fillet** .
The **Fillet** PropertyManager appears.
- Enter **10** for the **Radius**.
- Select **Full preview**.
Leave the remaining settings at their default values.






- 4 Click the first corner edge.

The faces, edges, and vertices are highlighted as you move the pointer over them.

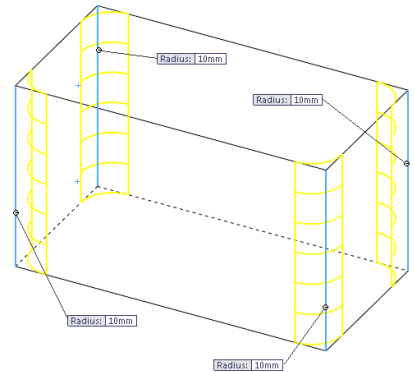
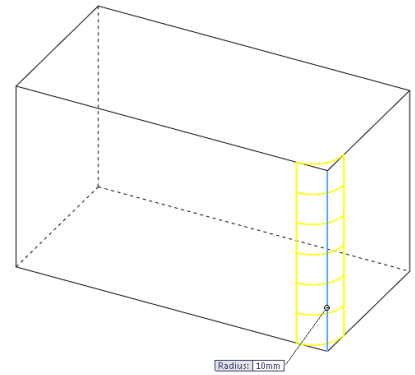
When you select the edge, a callout **Radius: 10mm** appears.

- 5 Identify selectable objects. Notice how the pointer changes shapes:

Edge:  Face:  Vertex: 

- 6 Click the second, third and fourth corner edges.

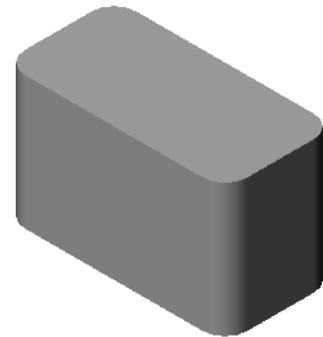
Note: Normally, a callout only appears on the *first* edge you select. This illustration has been modified to show callouts on each of the four selected edges. This was done simply to better illustrate which edges you are supposed to select.



- 7 Click **OK** .

Fillet1 appears in the FeatureManager design tree.

- 8 Click **Display Style > Shaded**  on the Heads-up View toolbar.



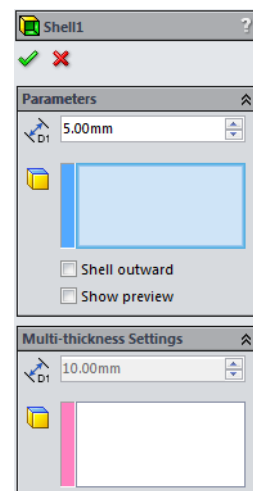
Hollow Out the Part

Remove the top face using the Shell feature.

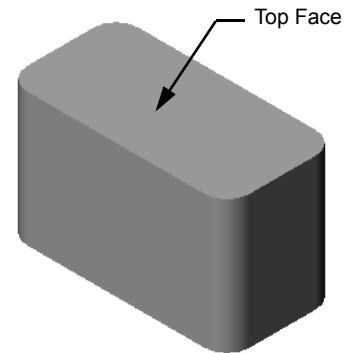
- 1 Click **Features > Shell** .

The **Shell** PropertyManager appears.

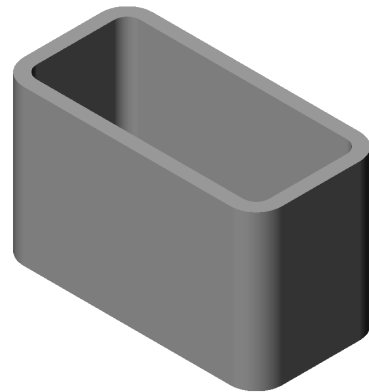
- 2 Enter **5** for **Thickness**.



- 3 Click the top face.



- 4 Click .





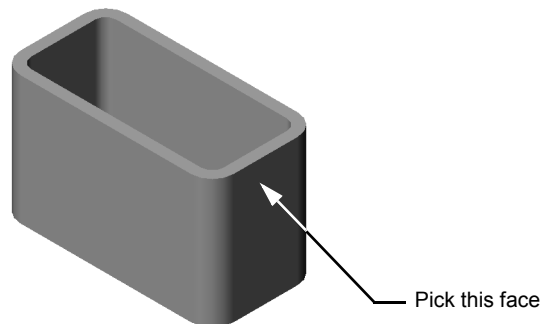
Extruded Cut Feature

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

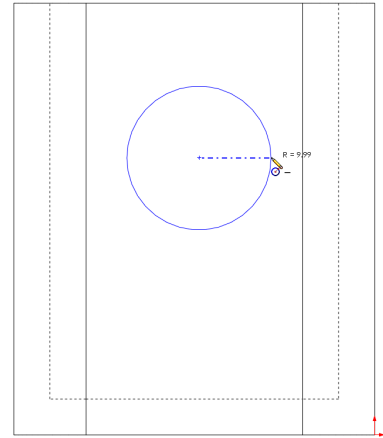
Open a Sketch

- 1 To select the sketch plane, click the right-hand face of the box.
- 2 Click **View Orientation > Right**  on the Heads-up View toolbar.
The view of the box turns. The selected model face is facing you.
- 3 Open a 2D sketch. Click **Sketch > Sketch** .




Sketch the Circle

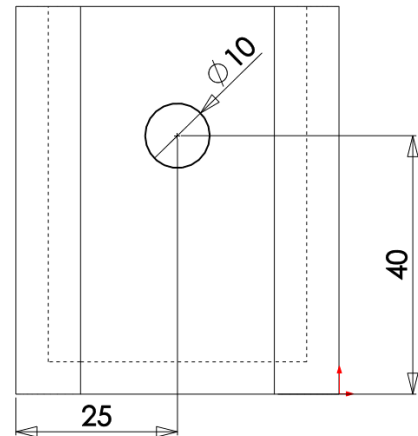
- 1 Click **Sketch > Circle** .
- 2 Position the pointer where you want the center of the circle. Click the left mouse button.
- 3 Drag the pointer to sketch a circle.
- 4 Click the left mouse button again to complete the circle.





Dimension the Circle

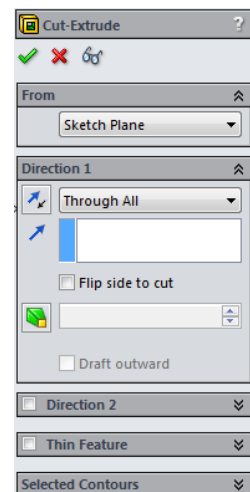
Dimension the circle to determine its size and location.

- 1 Click **Sketch > Smart Dimension** .
- 2 Dimension the diameter. Click on the circumference of the circle. Click a location for the dimension text in the upper right corner. Enter **10**.
- 3 Create a horizontal dimension. Click the circumference of the circle. Click the left most vertical edge. Click a location for the dimension text below the bottom horizontal line. Enter **25**.
- 4 Create a vertical dimension. Click the circumference of the circle. Click the bottom most horizontal edge. Click a location for the dimension text to the right of the sketch. Enter **40**.



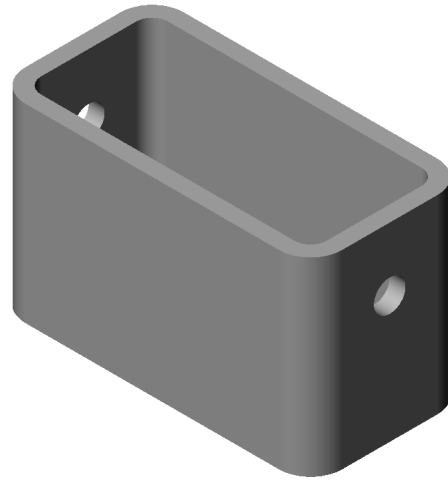
Extrude the Sketch

- 1 Click **Features > Extruded Cut** . The **Extrude** 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.

- 1 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.
- 2 Click **View Orientation > Isometric**  on the Heads-up View toolbar.

Save the Part

- 1 Click **Save**  on the Menu Bar.
- 2 Click **File, Exit**.

Lesson 2 — 5 Minute Assessment

Name: _____ Class: _____ Date: _____

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

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?

Exercises and Projects — Designing a Switch Plate

Switch plates are required for safety. They cover live electrical wires and protect people from electric shock. Switch plates are found in every home and school.

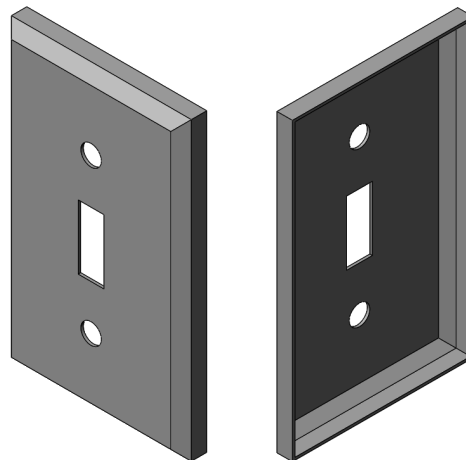
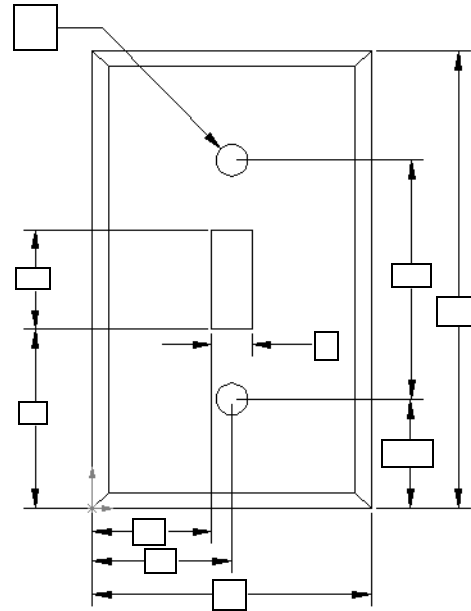
 **Caution:** Do not use metal rulers near switch plates attached to a live wall outlet.

Tasks

- 1 Measure a single light plate switch cover.

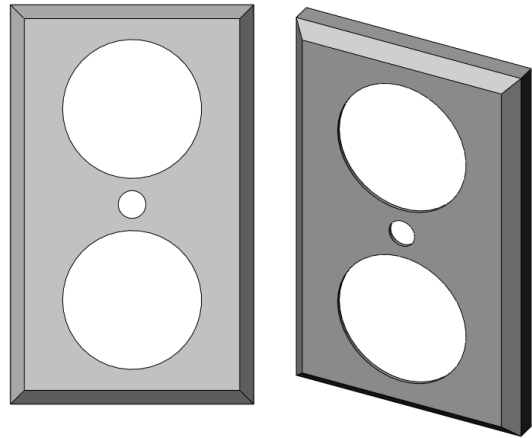
- 2 Using paper and pencil, manually sketch the light plate switch cover.
- 3 Label the dimensions.
- 4 What is the base feature for the light plate switch cover?

- 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?



Lesson 2: Basic Functionality

- 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

Name: _____ Class: _____ Date: _____

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

- 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: _____