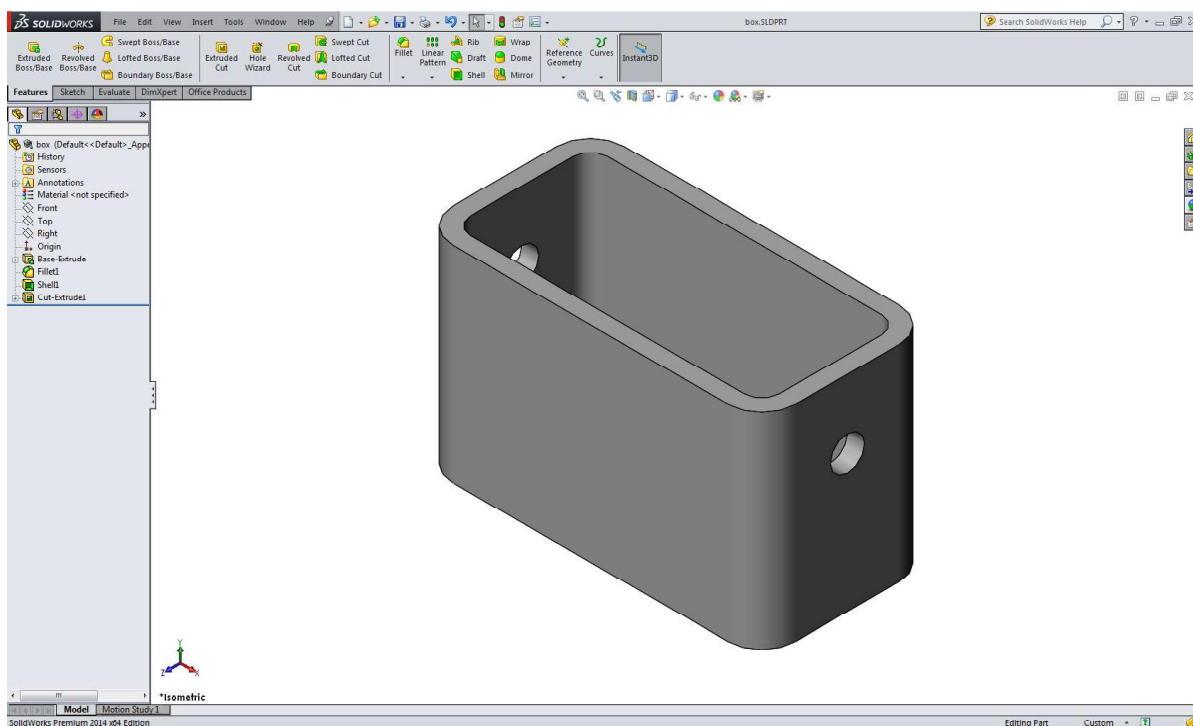


## Lesson 2: Basic Functionality

### 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](http://www.solidworks.com/students)

## Competencies for Lesson 2

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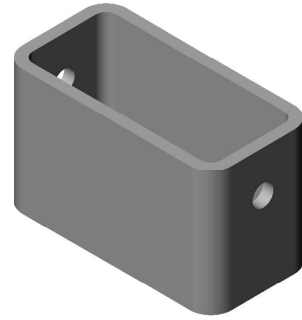
You develop the following competencies in this lesson:

- ❑ **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.

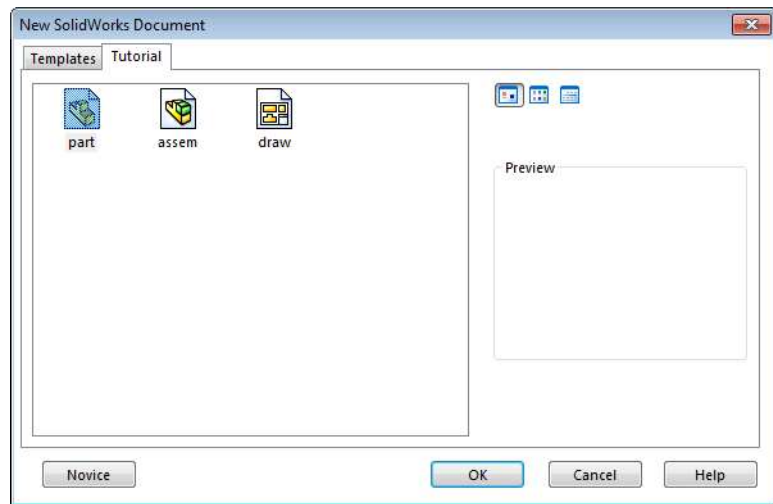
The step-by-step instructions are given below.



### Create a New Part Document

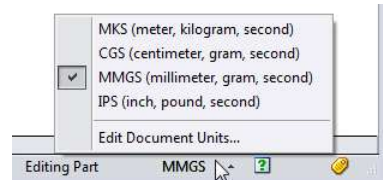
- 1 Create a new part. Click **New**  on the Menu Bar.
- 2 Click the **Tutorial** tab.
- 3 Select the **Part** icon.
- 4 Click **OK**.

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**.

### 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.

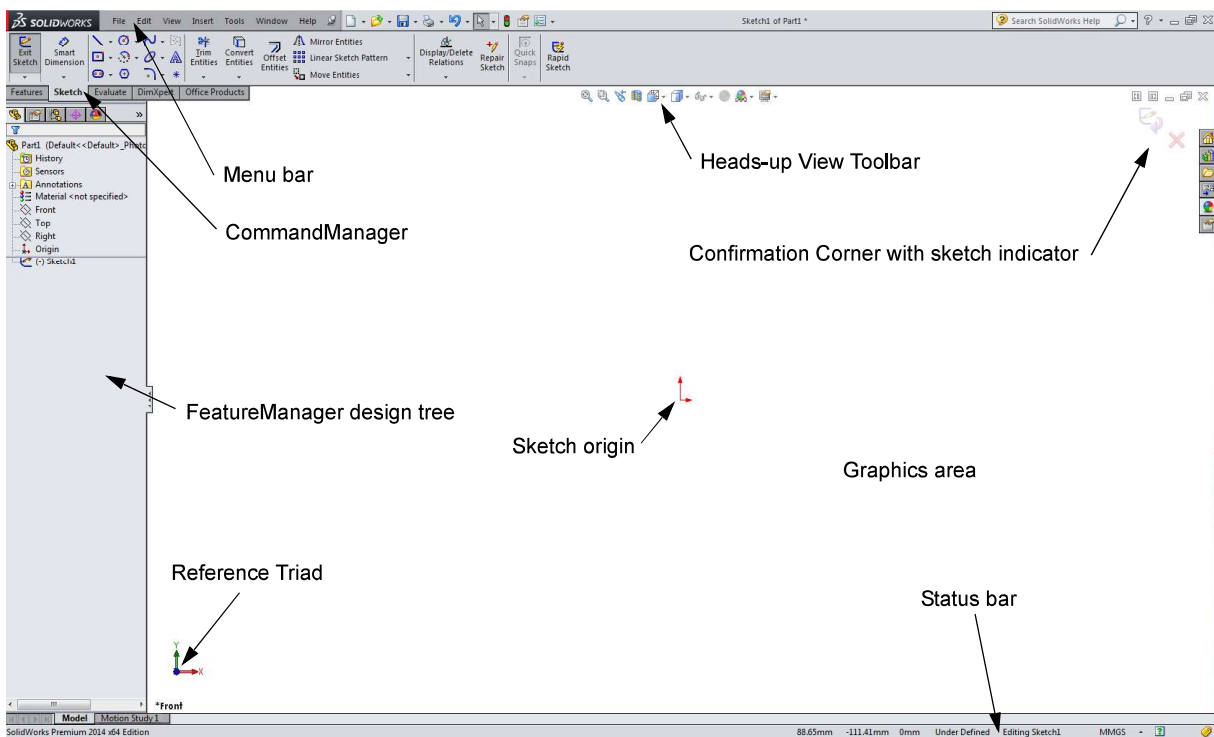


When other commands are active, the confirmation corner displays two symbols: a check mark and an X. The check mark executes the current command. The X cancels the command.




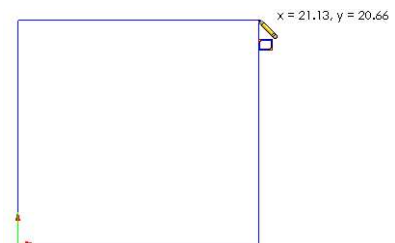
### Overview of the SolidWorks Window

- 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.







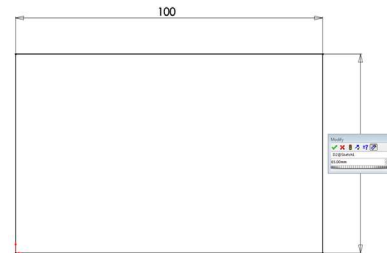
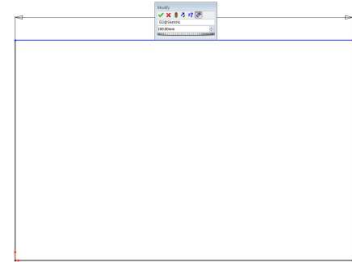
### Sketch a Rectangle

- 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.

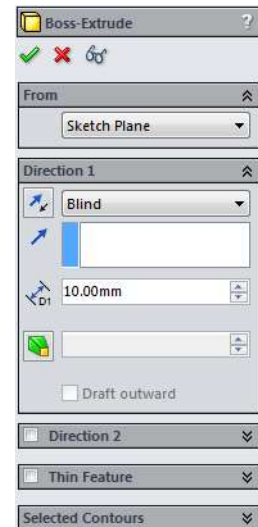
- 1 Double-click **65**.  
The **Modify** dialog box appears.
- 2 Enter **60** in the **Modify** dialog box.
- 3 Click .



## 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.

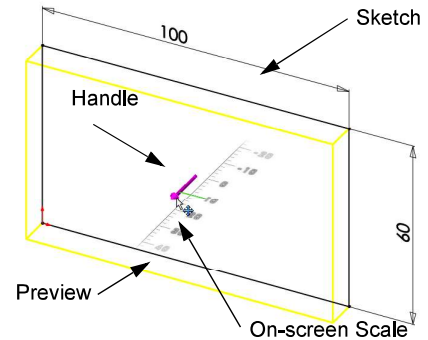
- 1 Click **Features > Extruded Boss/Base** .  
The **Boss-Extrude** PropertyManager appears. The view of the sketch changes to trimetric.




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.

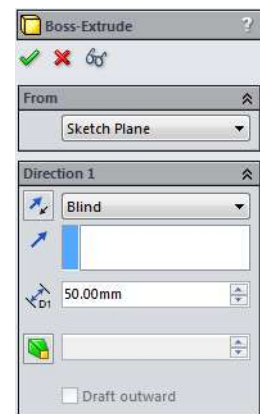


The cursor changes to . If you want to create the feature now, click the right mouse button. Otherwise, you can make additional changes to the settings. For example, the depth of extrusion can be changed by dragging the dynamic handle with the mouse or by setting a value in the PropertyManager.

3 Extrude feature settings.

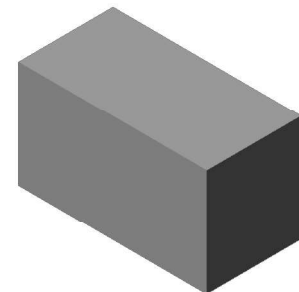
Change the settings as shown.

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



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

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



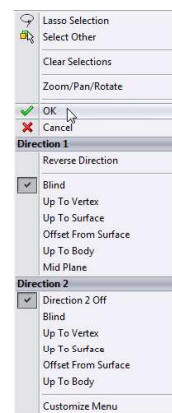
**TIP:**


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

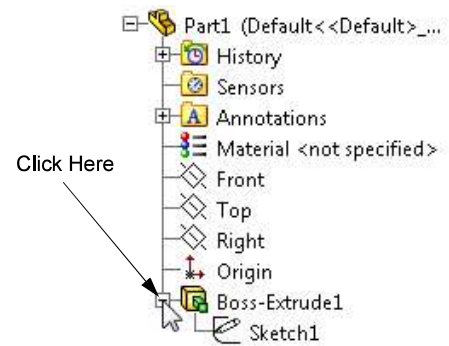
A second method is the set of **OK/Cancel** buttons in the confirmation corner of the graphics area.




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



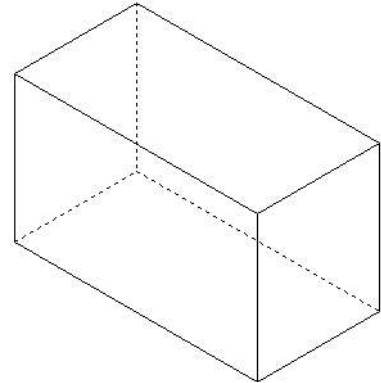
- Click the plus sign  beside **Boss-Extrude1** in the FeatureManager design tree. Notice that **Sketch1** — which you used to extrude the feature — is now listed under the feature.



### 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.

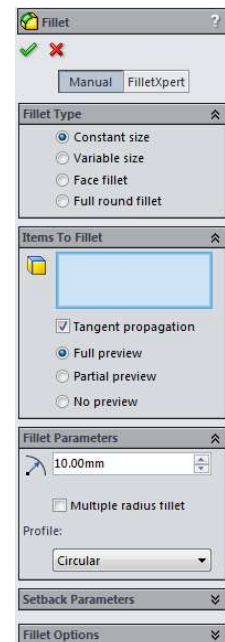
- Click **Features > Fillet** .

The **Fillet** PropertyManager appears.

- Enter **10** for the **Radius**.

- Select **Full preview**.

Leave the remaining settings at their default values.



## Lesson 2: Basic Functionality

- 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:

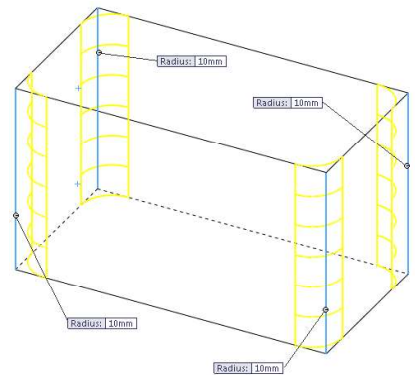
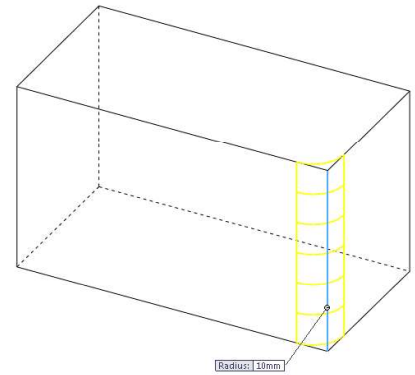
Edge:  Face:  Vertex: 

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

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**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.

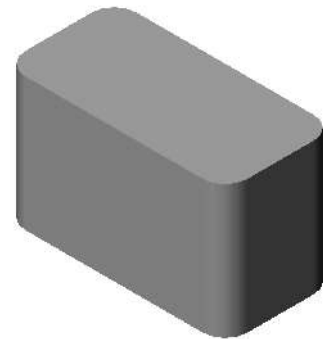
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- 7 Click **OK** .

**Fillet1** appears in the FeatureManager design tree.

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



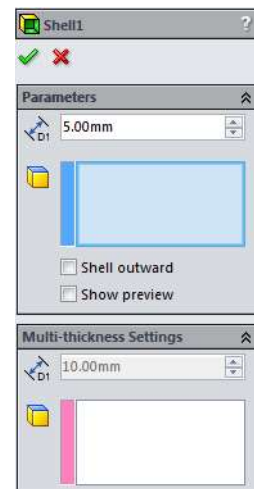
## Hollow Out the Part

Remove the top face using the Shell feature.

- 1 Click **Features > Shell** .

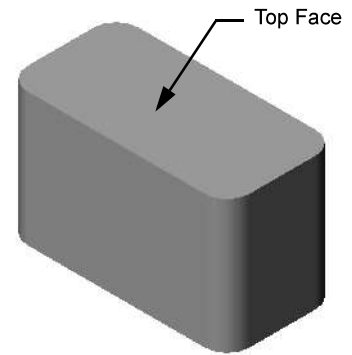
The **Shell** PropertyManager appears.

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

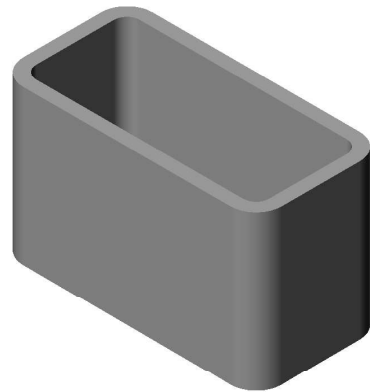




- 3 Click the top face.



- 4 Click .





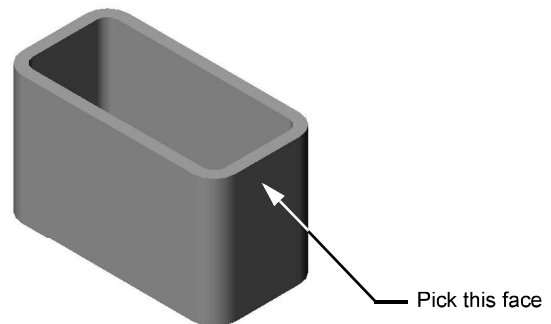
### Extruded Cut Feature

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

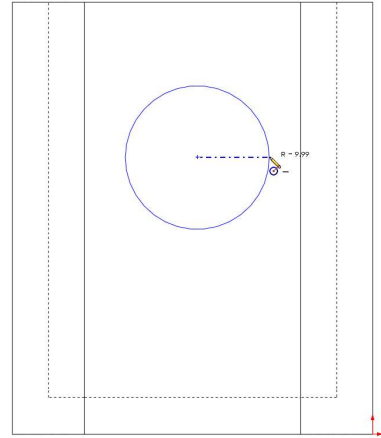
### Open a Sketch

- 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

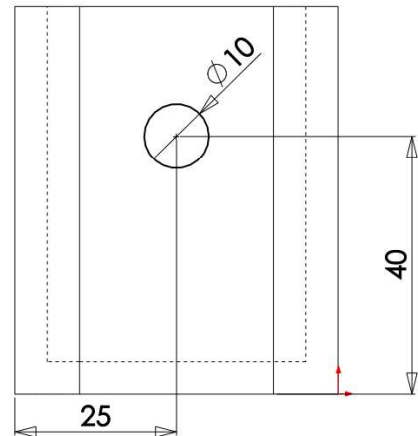
- 1 Click **Sketch > Circle** .
- 2 Position the pointer where you want the center of the circle. Click the left mouse button.
- 3 Drag the pointer to sketch a circle.
- 4 Click the left mouse button again to complete the circle.





### Dimension the Circle

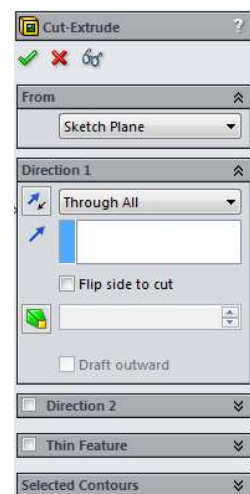
Dimension the circle to determine its size and location.

- 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**.



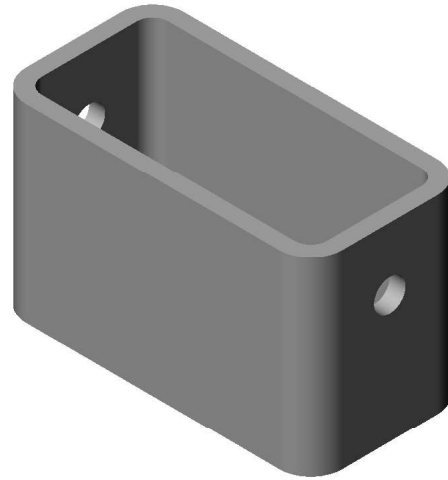
### Extrude the Sketch

- 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

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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.

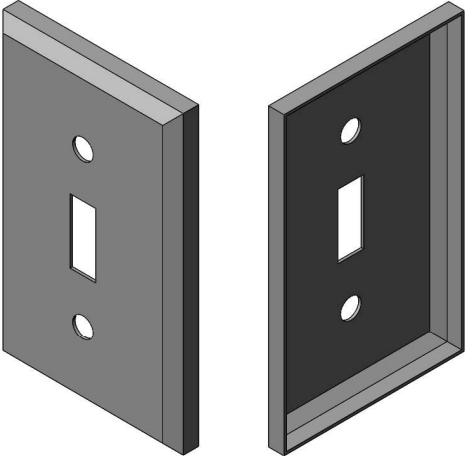
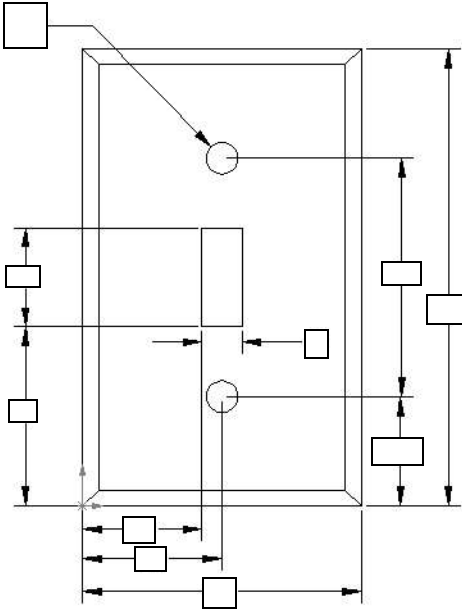
4 What is the base feature for the light plate switch cover?

\_\_\_\_\_  
\_\_\_\_\_  
\_\_\_\_\_  
\_\_\_\_\_

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

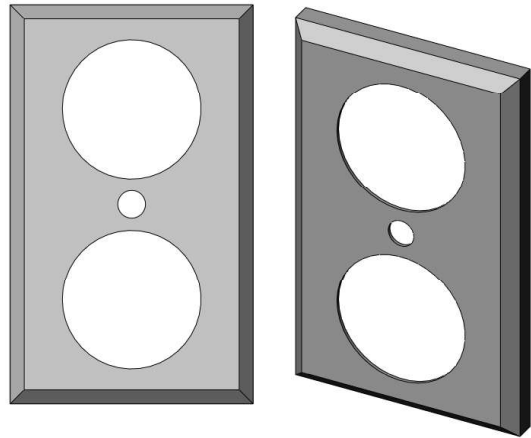
6 What features are used to develop the switchplate?

\_\_\_\_\_  
\_\_\_\_\_  
\_\_\_\_\_  
\_\_\_\_\_  
\_\_\_\_\_  
\_\_\_\_\_



**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 ( $90^\circ$ ) 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: \_\_\_\_\_