Engr 210 – **Engineering Graphics**
Lab 20: SolidWorks Basic Functionality

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. The rightmost panel is the Graphics Area, where you create and manipulate the part, assembly, or drawing.

Toolbars

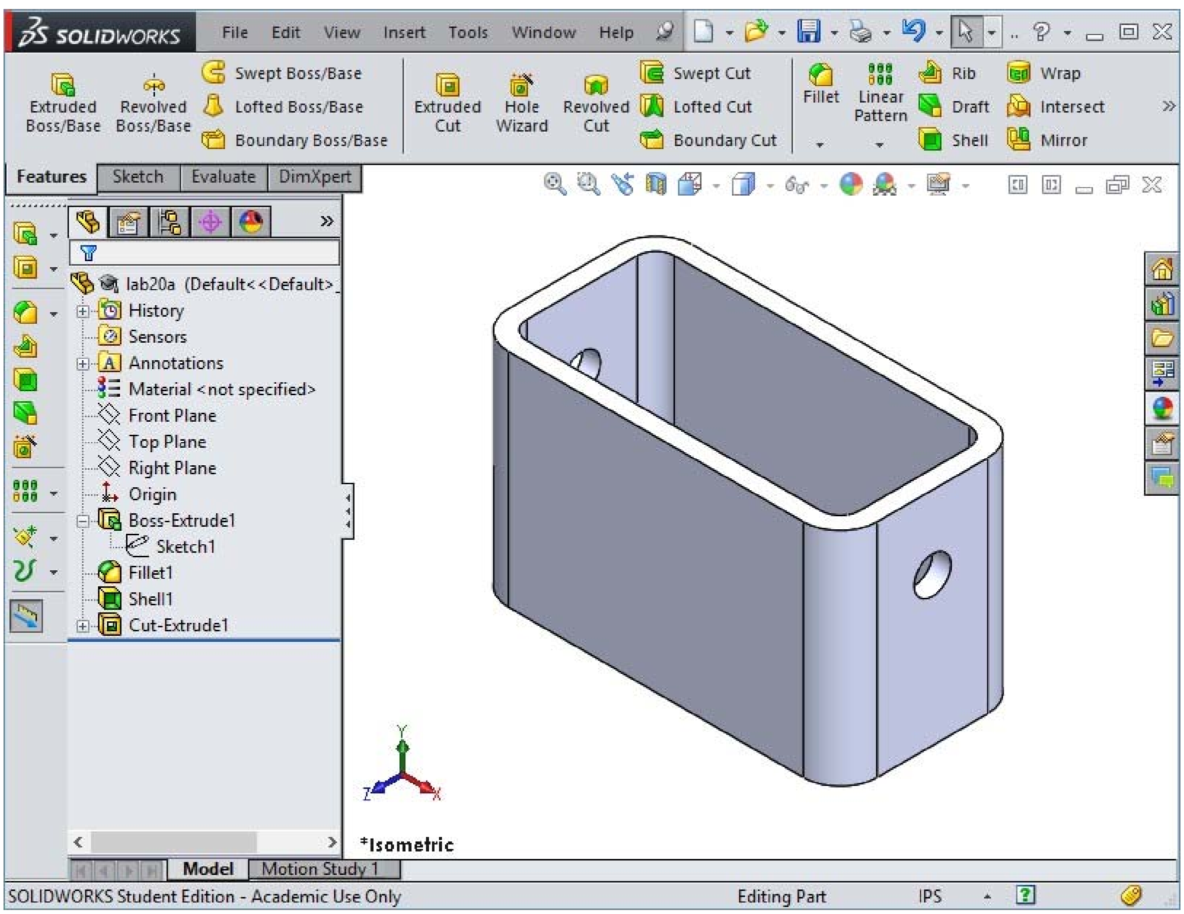
Toolbar buttons are shortcuts for frequently used commands. You can set toolbar placement and visibility based on the document type (part, assembly, or drawing). SolidWorks remembers which toolbars to display and where to display them for each document type.

CommandManager

The CommandManager is a context-sensitive toolbar that dynamically updates based on the toolbar you want to access. By default, it has toolbars embedded in it based on the document type.

When you click a button in the control area, the CommandManager updates to show that toolbar. Use the CommandManager to access toolbar buttons in a central location and to save space for the graphics area.

Let us use the basic SolidWorks commands to create the following part:

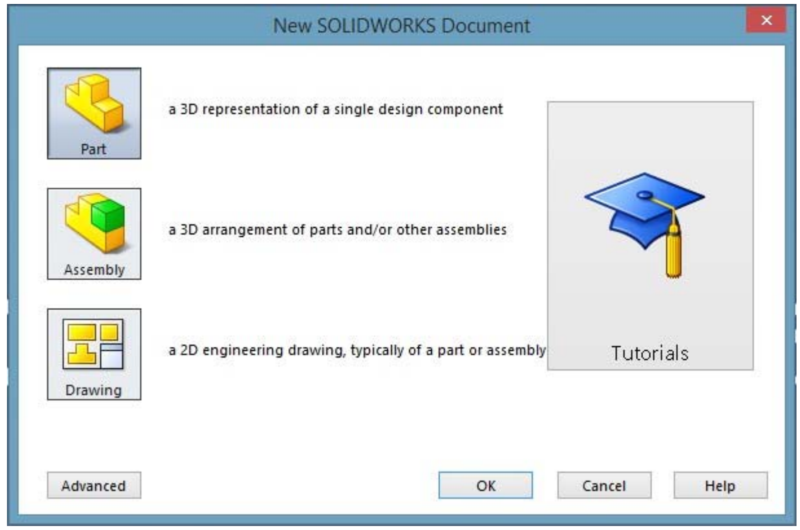


Part A. Create a New Part Document

1. Create a new part. Click on the Standard  toolbar. The New SolidWorks Document dialog box appears.

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

   A new part document window appears.



Base Feature

The Base feature requires:

- Sketch plane – ***`Front`*** (default plane)
- Sketch profile – 2D Rectangle
- Feature type – Extruded boss feature

Open a Sketch

1. Open a 2D sketch. Click  on the Sketch toolbar. Move the pointer over the Front plane, and click to select it.

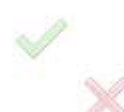
Confirmation Corner

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.

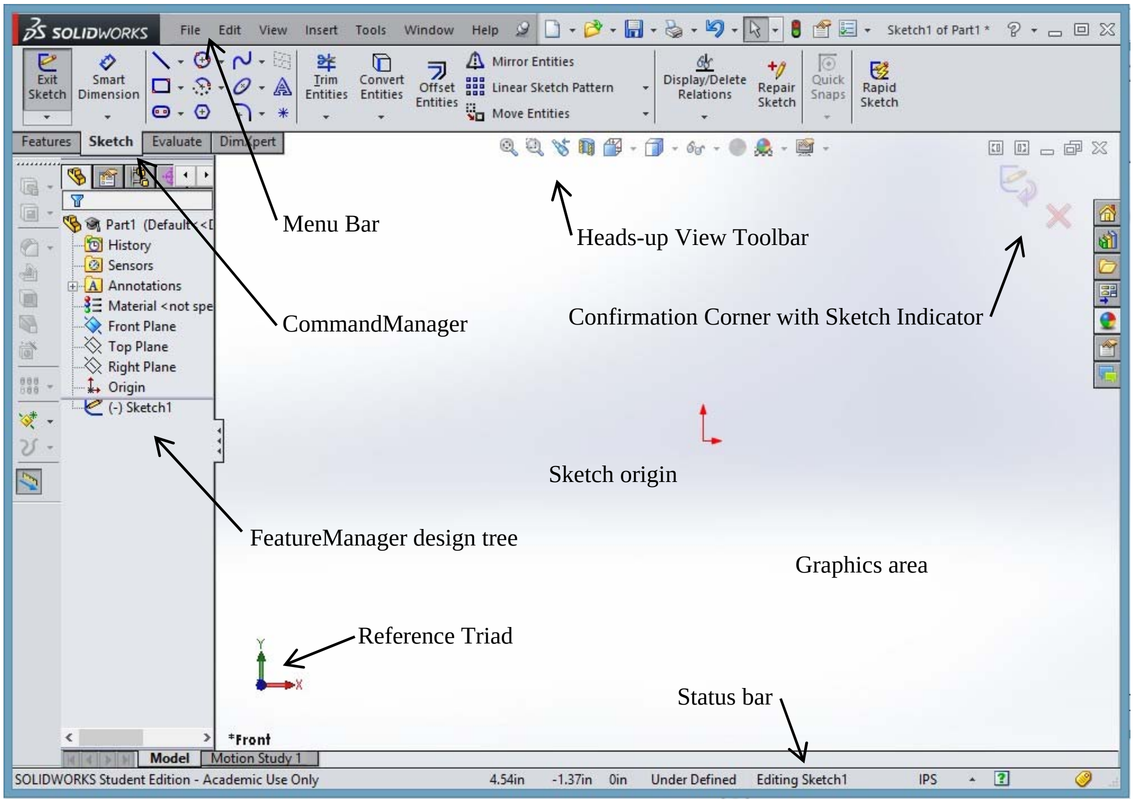
When other commands are active, the confirmation corner displays two symbols: a check mark and an X. The check mark executes the current command. The X cancels the command.



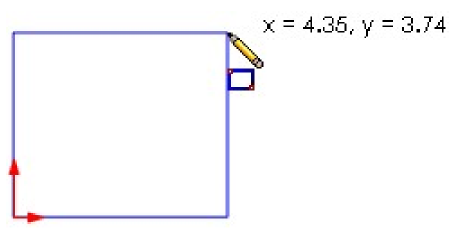
Overview of the SolidWorks Window

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

- 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



1. Click  on the Sketch Tools toolbar.
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 Smart Dimension  on the Sketch 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 ✓ or press **Enter**.

   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 ***part*** are 100mm x 60mm. Change the dimensions. Use the **Select** tool.

1. Double-click the dimension that was set to 65. The **Modify** dialog box appears.
2. Enter **60** in the **Modify** dialog box.
3. Click ✓ or press **Enter**.

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 **Extruded Boss/Base** on the Features toolbar. The ***Extrude Feature Property Manager*** appears. The view of the sketch changes to isometric.
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.

   Click **Detailed Preview** in the Property Manager to see the feature in shaded preview mode. Click on the icon again to go back to the previous view.

   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.

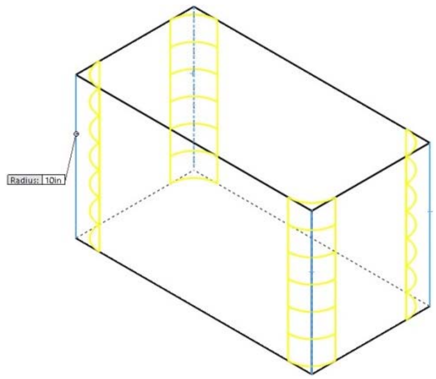
   Change settings to: End Condition = **Blind**; Depth = **50.**
4. Click ✓ or press **Enter** to create extrusion.

   The new feature, **`Extrude1`**, is displayed in the FeatureManager design tree.
5. 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.

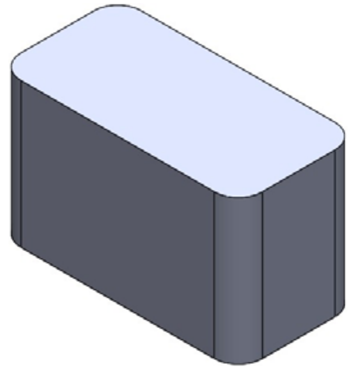
Fillet the Corners of the Part

1. Click **Hidden Lines Visible** on the **View** toolbar. This allows you to select hidden back edges of the part.
2. Click **Fillet** on the Features toolbar. The **Fillet** PropertyManager appears.
3. Enter **10** for the **Radius**. Leave the remaining settings at their default values.

4. Select the 4 vertical edges of the part.
5. Click  or press **Enter.** **`Fillet1`** appears in the FeatureManager design tree.



Before Fillet

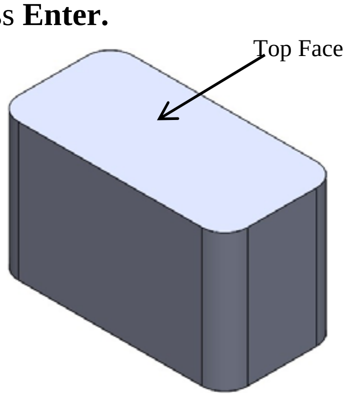


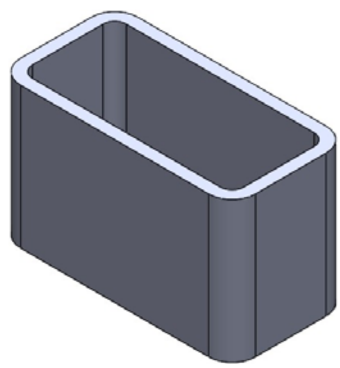
After Fillet

Hollow Out the Part

Remove the top face using the Shell feature.

1. Click  on the Features toolbar. The **Shell Feature** PropertyManager appears.
2. Enter **5** for **Thickness**.
3. Click the top surface.
4. Click  or press **Enter.**



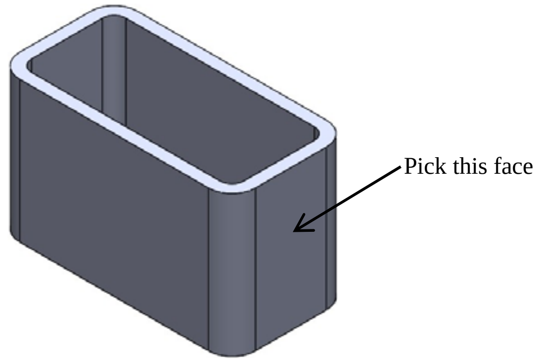


Extruded Cut Feature

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

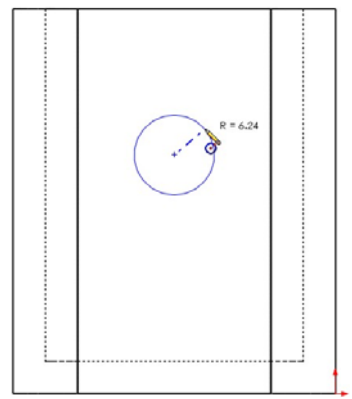
- Sketch plane – In this exercise, the face on the right-hand side of the part.
- Sketch profile – 2D circle

Open a Sketch



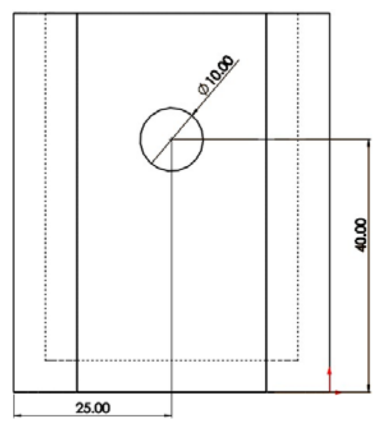
1. To select the sketch plane, click the right-hand face of the ***part***.
2. Click on the Standard Views toolbar, and select the *Normal to* option . The view of the ***part*** turns. The selected model face is facing you.
3. Open a 2D sketch. Click on the Sketch toolbar.

Sketch the Circle



1. Click on the Sketch Tools toolbar.
2. Position the pointer where you want the center of the circle. Click the left mouse button.
3. Drag the pointer to sketch a circle.
4. Click the left mouse button again to complete the circle.

Dimension the Circle

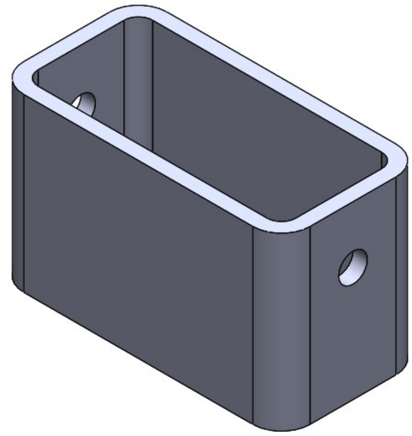


Dimension the circle to determine its size and location.

1. Click on the Sketch Relations toolbar.
2. 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. Enter **25**.
4. Create a vertical dimension. Click the circle and the bottom most horizontal edge. Click a location for the dimension text. Enter **40**.

Extrude the Sketch

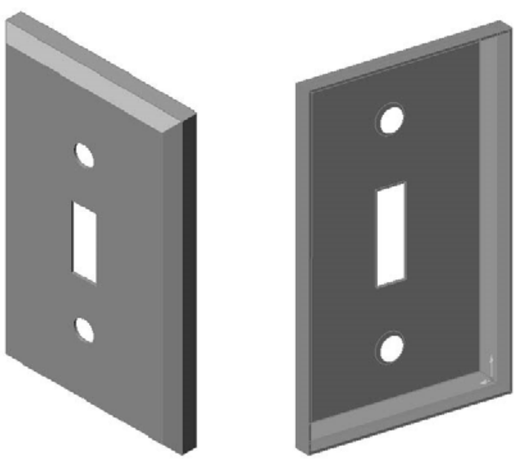
Dimension the circle to determine its size and location.

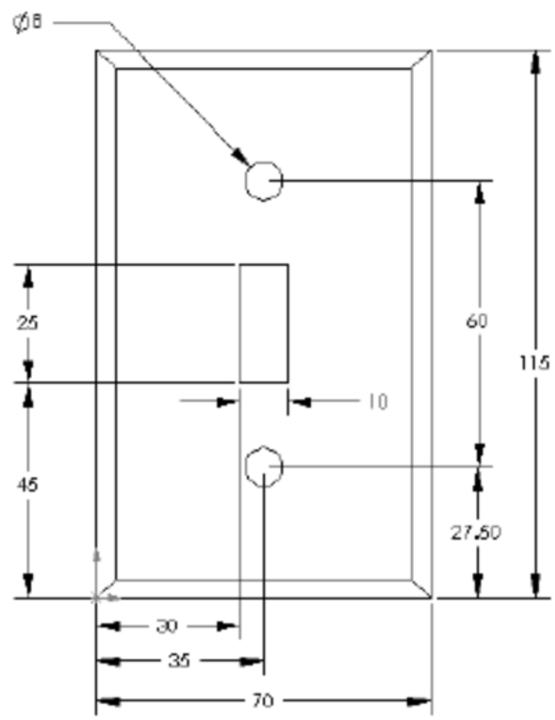


1. Click  on the Features toolbar. The **Extrude Cut Feature** PropertyManager appears.
2. Select **Through All** for the end condition.
3. Click  or press **Enter.**
4. The result is displayed.
5. Rotate the view in the graphics area to display the model from different angles by pressing and holding the middle mouse button, and dragging the pointer up/down or left/right.
6. Display the Isometric view by clicking  on the Standard Views toolbar.
7. Save the part as **Lab20a**.

Part B. Switch Plate

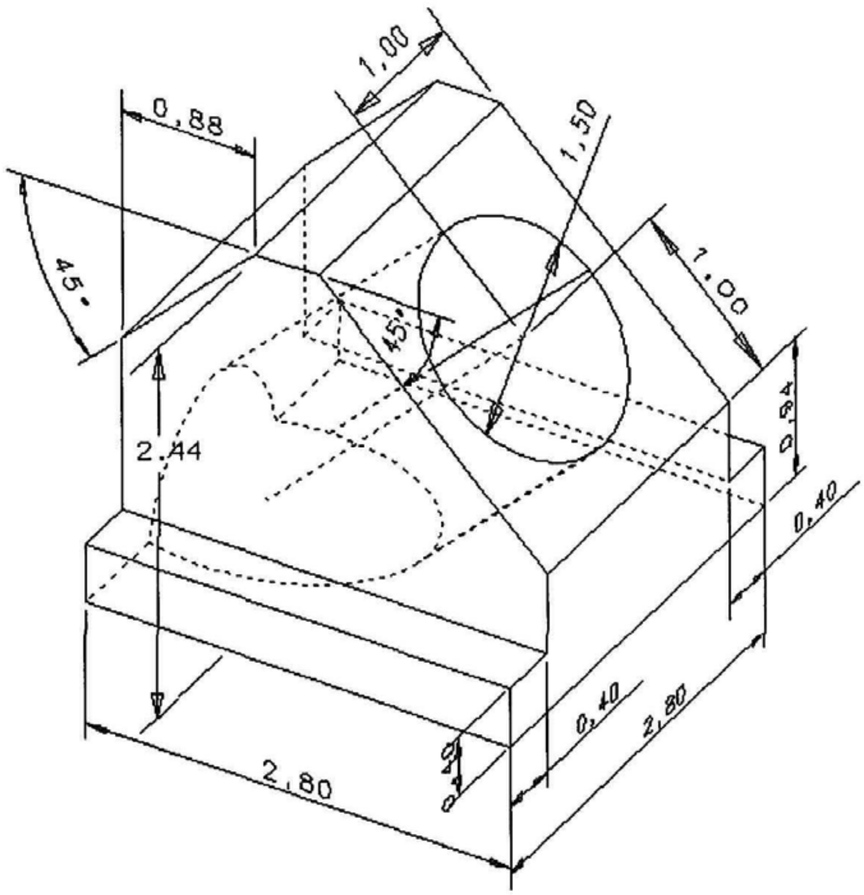
Create a SolidWorks part file of the switch plate whose dimensions are as shown. Assume reasonable values for missing dimensions (e.g., 10 mm for plate thickness). Save this file as **Lab20b**. Use the following SolidWorks commands: 1. Extrude boss/base, 2. Chamfer, 3. Shell, 4. Extrude cut.





Part C.

Create a SolidWorks part file of the given solid whose dimensions are as shown. Save this file as **Lab20c**. Use the following SolidWorks commands: 1. Extrude boss/base, 2. Chamfer, 3. Shell, 4. Extrude cut.



Part D. Online Tutorial: Lesson 1

Follow the instructions in **Lesson 1 – Parts** of the SolidWorks Tutorial. Lesson 1 can be found in SolidWorks Resource → Tutorials → Getting Started → Lesson 1.