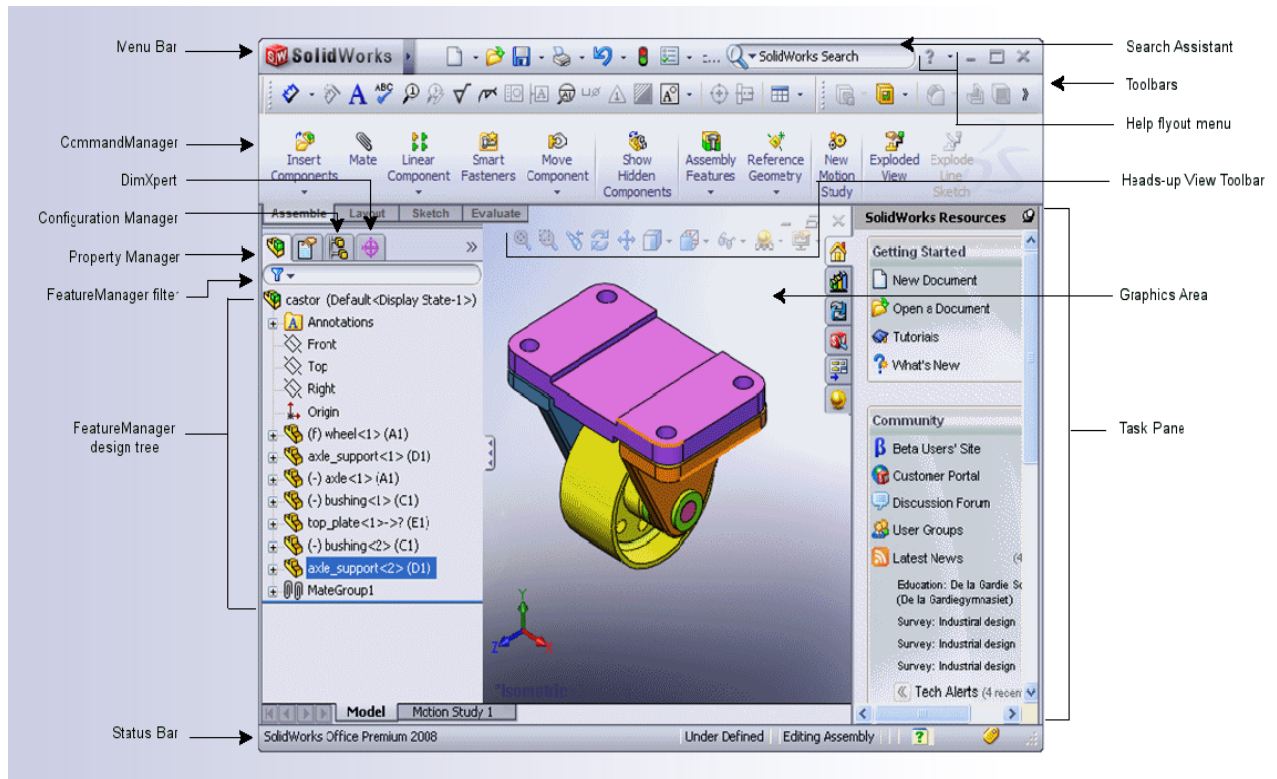


This tutorial will create a box with round and rectangular holes, a top for the box, and a dimensioned drawing of the box.

- **Bold words** imply selection of a menu command or icon. Menu item location is indicated in parenthesis after each command.
- The following graphic will be useful for finding the necessary SolidWorks commands. This graphic may be found in SolidWorks by selecting **User Interface Overview** in the help menu contents.
- Mouse clicks are done with the left button unless otherwise noted.



1. Create the box top

- **Document** icon – **New – Part** (Menu Bar)
- **Save** icon – **Save As...** – “**Firstname Lastname Lab01 Sec## Top.SLDPRT**” (fill in your name and the appropriate digits for the # characters) (Menu Bar)
- **Tools** icon - **Options - Document Properties** (Menu Bar)
 - **Units**
 - **Length Decimals** .123 (Property Manager)
 - **Unit System IPS**
- **Sketch – Rectangle** (Command Manager) - select top plane
 - **View Orientation** icon (Heads-up View Toolbar) – **Normal To**
 - Create a rectangle with the origin placed at the bottom left corner of the rectangle. Do not be concerned about the size of the rectangle.
- **Smart Dimension** (Command Manager)
 - Select the top of the rectangle, drag the dimension text to the desired location and enter 4.6".
 - Select the left side of the rectangle, drag the dimension text to the desired location and enter 3".
 - Note that all lines have changed color from blue to black. This indicates the sketch is fully defined.
 - Select the right side of the rectangle, drag the dimension text to the desired location and enter 3". Note what happens.
- Select the **Zoom To Fit** icon (Heads-up View Toolbar).
- Close the sketch by selecting the arrow/pencil icon in the upper right corner of the drawing window.
- **Features - Extruded Boss/Base** (Command Manager)
 - Enter **0.25"** in the **D1** (Property Manager) window at the left side of the screen and select the green checkmark.
- Click and hold the mouse scroll button while moving the mouse or select a **View Orientation** icon (Heads-up View Toolbar) to look at a variety of orientations of the part.
- Save the model.

2. Create the box

- **Document** icon – **New – Part** (Menu Bar)
- **Save** icon – **Save As...** – “**Firstname Lastname Lab01 Sec## Box.SLDPRT**” (fill in your name and the appropriate digits for the # characters) (Menu Bar)
- **Tools** icon - **Options - Document Properties** (Menu Bar)
 - **Units - Decimal places** = 3 (Property Manager)
- **Sketch - Rectangle** (Command Manager) - select top plane
 - **View Orientation** icon (Heads-up View Toolbar) – **Normal To**
 - Create a rectangle with the origin at the bottom left. Do not be concerned about the size of the rectangle.
- **Smart Dimension** (Command Manager)
 - Select the top of the rectangle, drag the dimension text to the desired location and enter 4.6".
 - Select the left side of the rectangle, drag the dimension text to the desired location and enter 3".
- Select the **Zoom To Fit** icon (Heads-up View Toolbar).
- Close the sketch by clicking the arrow/pencil icon in the upper right corner of the graphics window.
- **Features - Extruded Boss/Base** (Command Manager)
 - Enter 1.5" in the **D1** (Property Manager) window and select the green checkmark.
- Set the view to **Top** (Heads-up View Toolbar).
- Click the top surface to select it.
 - Select the **Sketch** tab
 - **Sketch** icon - **Offset** (Command Manager)
 - Enter 0.125" in the **Offset Distance D** (Property Manager) window
 - Select the **Reverse** option (Property Manager) to force the offset to the inside of your existing Box and select the green checkmark.
- **Sketch Fillet** (Command Manager) icon
 - Set the **Fillet Parameters** to **0.13"** (Property Manager)
 - Sequentially select each corner of the offset rectangle to create a radius at all corners.
 - Select the green checkmark.
- **Features - Extruded Cut** (Command Manager)
 - In the direction window select **Through All** (Property Manager).
 - Rotate the model to see a preview of the result.
 - Change direction type to **Blind** and set **D1** to **1.375"** (Property Manager)
 - Select the green checkmark.
- Orient the view perpendicular to the front of the box and select the front.

- **Sketch - Circle** (Command Manager)
 - Draw a circle on the front.
 - Use **Smart Dimension** (Command Manager) to set the circle diameter to **0.5"**.
 - Use **Smart Dimension** (Command Manager) to select the circle and the left edge of the box front to place the circle **1"** from the left side of the box.
 - Use **Line – Centerline** (Command Manager) to draw a centerline horizontally across the front of the box. Note how the cursor will snap to the center of each vertical edge of the box front.
 - Expand **Display/Delete Relations** and select **Add Relations** (Command Manager).
 - Select the center of the hole and the centerline.
 - Click on **Coincident** in the **Add Relations** (Property Manager) window on the left of the screen and select the green checkmark.
- **Features - Extruded Cut** (Command Manager)
 - In the direction window select **Up To Next** (Property Manager).
 - Rotate the model to see a preview of the result.
 - Select the green checkmark.
- **Features - Linear Pattern** (Command Manager)
 - Select the top front edge of the box such that the edge is indicated in the highlighted blue window under **Direction 1** (Property Manager).
 - Select the inside surface of the extruded hole – do not select the inside surface of the box.
 - Set **D1** to 1.2" (Property Manager).
 - Change **Number of Instances** to 3 (Property Manager).
 - If the holes are copied in the wrong direction, reverse the direction vector by clicking on the blue and black arrow icon adjacent to the **Direction 1** window
 - Select the green checkmark.
- Use the mouse or the **Standard Views** (Heads-up View Toolbar) tool bar to look at a variety of orientations of the part.
- Save the model.

3. Assemble the Box

- **Document** icon – **New – Assembly** (Menu Bar)
 - **Begin Assembly – Part/Assembly to Insert – Box** (Property Manager)
 - Click to place Box in model window
- **Save** icon – **Save As...** – “**Firstname Lastname Lab## Sec## BoxAssembly.SLDASM**” (fill in your name and the appropriate digits for the # characters) (Menu Bar)
- **Insert Components - Part/Assembly to Insert – Top** (Property Manager)
 - Click to place Top above box
- **Mate** icon (Command Manager)
 - Select front of Top, select front of Box and click green checkmark to align the top with the box.
 - Select a side of Top, select a side of Box and click green checkmark to mate
 - Use the mouse to move box top. Notice how it is constrained to move vertically
 - Select bottom of Top, select top surface of Box and click green checkmark to mate
- Save the assembly

4. Create a Drawing

- **Open** College Shop 8.5x11 Template.SLDDRW (Menu Bar)
- **File** icon – **Save As...** – “**Firstname Lastname Lab## Sec## BoxDrawing.SLDDRW**” (fill in your name and the appropriate digits for the # characters) (Menu Bar)
- **View Layout/Model View** (Command Manager)
 - In the **Part/Assembly to Insert - Open documents** (Property Manager) window select **Box**.
 - Click the blue **Next** (Property Manager) arrow.
 - In the **Orientation** (Property Manager) window select the icon representing the **Front** view.
 - Move the cursor to the center of the screen and click.
 - Move the cursor to the left of the front view and click.
 - Move the cursor above the front view and click.
 - Move the cursor to the upper right of the front view and click (to create an isometric view).
 - Select the green checkmark (Property Manager).
- Drag the views to space them appropriately.
- Double click on the title block text to edit with appropriate information.
- Select the front view, **Display Style** icon (Graphics Area) and click on the **Hidden Lines Visible** icon.
- Select the isometric view and change the **Display Style** (Graphics Area) to **Shaded With Edges**
 - Delete extraneous centerlines on the isometric view
- **Annotations – Center Mark** (Command Manager)
 - Click on the left hole and use the **Propagate** icon to extend center lines across all three holes.
- **Sketch – Centerline**
 - Create centerlines across the top of the box.
- **Annotations – Notes** (Command Manager)
 - Select the intersection of the top centerlines and place the text to the right of the box.
 - Edit the text to read 4.350 x 2.750 x 1.375.
- **Smart Dimension** (Command Manager)
 - Dimension remaining as shown on the example on the next page.
 - As necessary, add additional text by selecting the dimension and editing its text in the **Dimension Text** window (Property Manager) on the left side of the screen.
- Save the drawing.