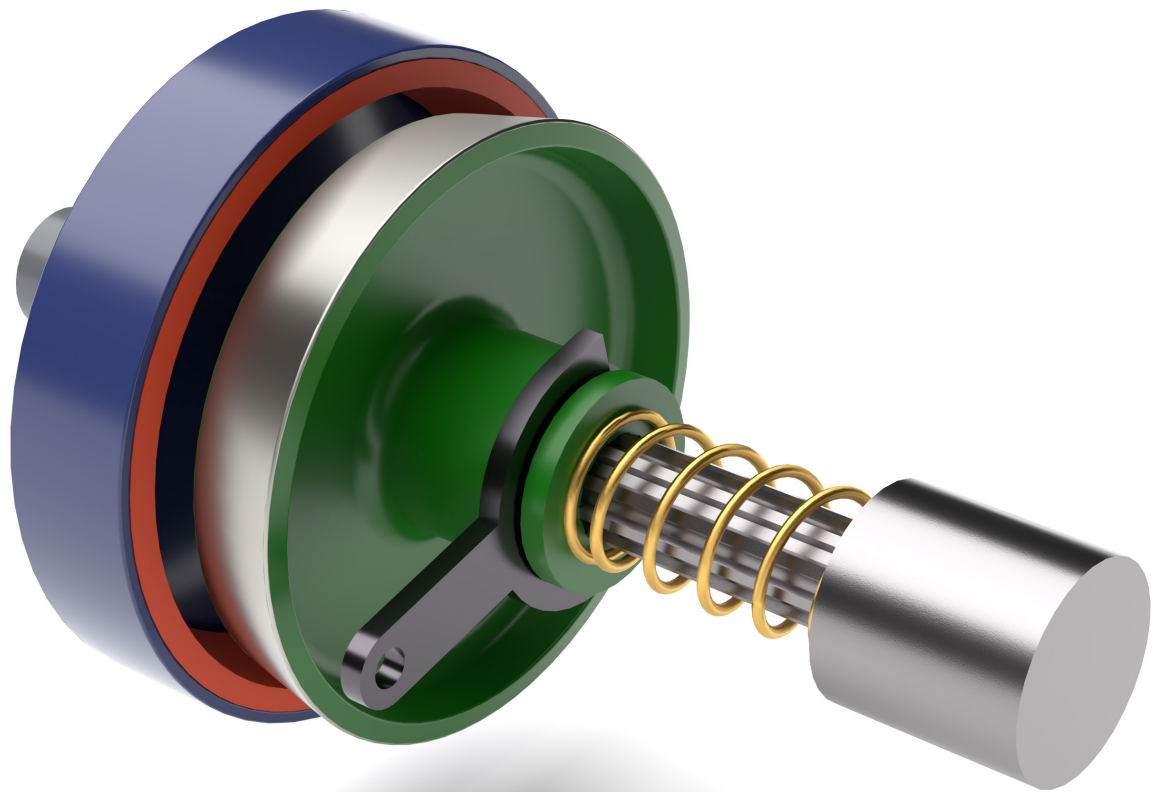# Designing with SOLIDWORKS® 2025

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Chapter 1. Getting Started

Introduction

This book will concentrate on the modeling application used to create solid models of parts, engineering design and drawings, and assemblies.

The tutorials in this book assume you are using the default settings of SOLIDWORKS in a university setting.

SOLIDWORKS uses the mouse and its three buttons extensively. Therefore, it is essential to understand the mouse button's basic functions listed below.

LMB—(LEFT MOUSE BUTTON)

Used for most operations such as selecting icons, picking graphic entities, and selecting menu items.

MMB—(MIDDLE MOUSE WHEEL)

Used to maneuver the entities in the window. To **Rotate** hold down the wheel button then move the mouse. To **Pan** hold down the wheel button along with the <Ctrl> key and move the mouse. To **Zoom** hold down the wheel button along with the <Shift> key and move the mouse. Spinning the wheel causes the entities to zoom in or out relative to the position of the mouse cursor.

RMB—(RIGHT MOUSE BUTTON)

Hold down the button to bring up additional available options in a context-sensitive pop-up menu.

The **<Esc>** key is used to cancel a command. Sometimes it may be necessary to press the **<Esc>** key twice to cancel a command.

The <**F**> key resizes the entities in the window to fit in the window. Entities typically go off-screen when they are being resized or moved.

The <**G**> key activates the magnifying glass feature so you can zoom in on an area.

<Z> key and Shift+<Z> key

Pressing the <Z> key zooms out. Holding down the <Shift> key and pressing the <Z> key zooms in.

Using just the Arrow keys

Pressing the arrow key rotates the entities on the screen left, up, down, or right.

Holding down the <Shift> key and pressing the arrow keys rotate the entities on the screen 90 degrees left, up, down, or right.

Holding down the <Alt> key and pressing the left arrow key rotates the entities counterclockwise. <Alt> key and right arrow key rotate the entities clockwise and normal to the screen.

Below is a list of other keystrokes used in SOLIDWORKS and their meaning.

<Ctrl> + <1>	makes the Front View normal to the screen
<Ctrl> + <2>	makes the Back View normal to the screen
<Ctrl> + <3>	makes the Left View normal to the screen
<Ctrl> + <4>	makes the Right View normal to the screen
<Ctrl> + <5>	makes the Top View normal to the screen
<Ctrl> + <6>	makes the Bottom View normal to the screen
<Ctrl> + <7>	makes the Isometric View normal to the screen
<Ctrl> + <8>	makes the current selected surface normal to the screen

Note: It is good practice to create a **SolidWorks Templates** folder to store your templates in and a **SolidWorks Parts** folder to store your data files in before starting SOLIDWORKS. When saving files in SOLIDWORKS, you should always verify that you are saving them in the proper folder before picking the **Save** or **Save As** button.

When working with SOLIDWORKS, it is strongly advised to ***save your work often*** in case of a power failure or an unexpected program crash.

> **Note that SOLIDWORKS 2025 is backward compatible with only SOLIDWORKS 2024 and 2023. SOLIDWORKS 2024 is still backward compatible with SOLIDWORKS 2023 and 2022.**

1. Before starting SOLIDWORKS, create a **SolidWorks Part** folder and a **SolidWorks Templates** folder on the desktop.

2. Double-click the left mouse button (LMB) on the SOLIDWORKS 2025 icon on the desktop to start SOLIDWORKS.

3. If SOLIDWORKS is in Novice mode, the following screen will appear.
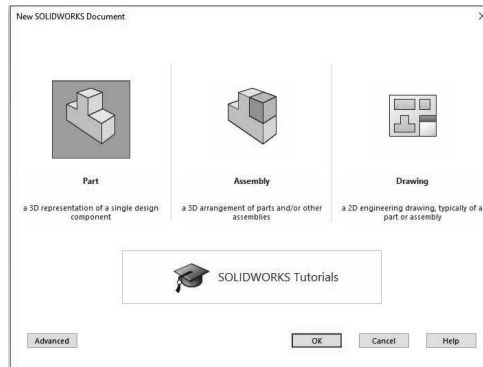
4. Pick the **Part** icon, then pick the **OK** button. The following screen will appear. Right-click in the menu area above the tabs to bring up a pop-down menu. Check the **Use Large Buttons with Text** option.
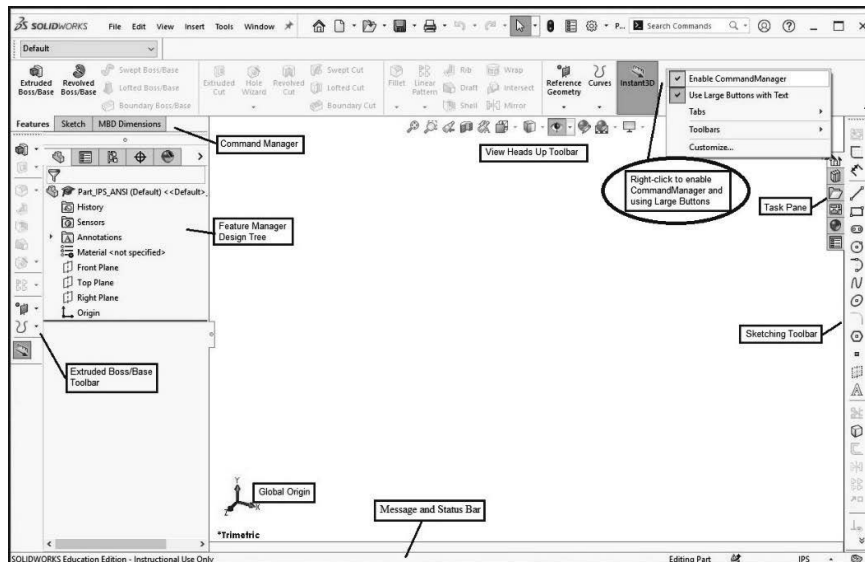
5. ⚙️ Pick the **Options** icon at the top of the window. When the System Options window appears, pick **Colors**. Then select **Viewport Background** in the Color Scheme settings area. Pick the **Edit** button, then select White from the color menu. Check the **Plain (Viewport Background color above)** option in the Background appearance area. Pick the **OK** button. This will produce a white background while you are in SOLIDWORKS, which is my preferred background color.

6. Change the **Top Gradient Color** and the **Bottom Gradient Color** to white as well. Pick **OK.**
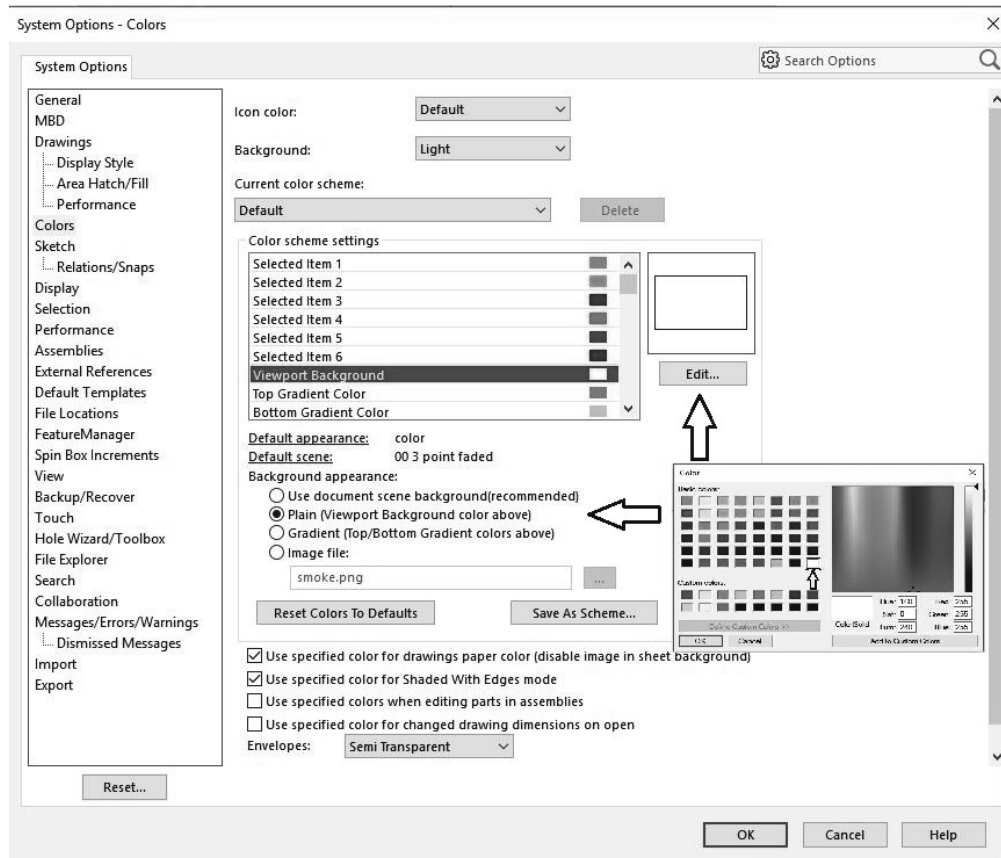
7. In the lower right corner of the screen, pick the unit's area (either IPS or MMGS), then pick Edit Document Units… Most of the exercises in this textbook use the IPS system so make sure IPS is selected.
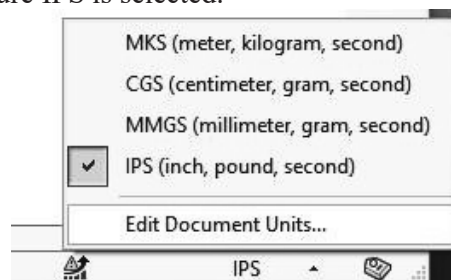
8. ⚙️ Pick the **Options** icon at the top of the window again. Pick the Document Properties tab. Set Overall Drafting Standard to ANSI and Length Units to 3-decimal places for inches, then pick **OK**.
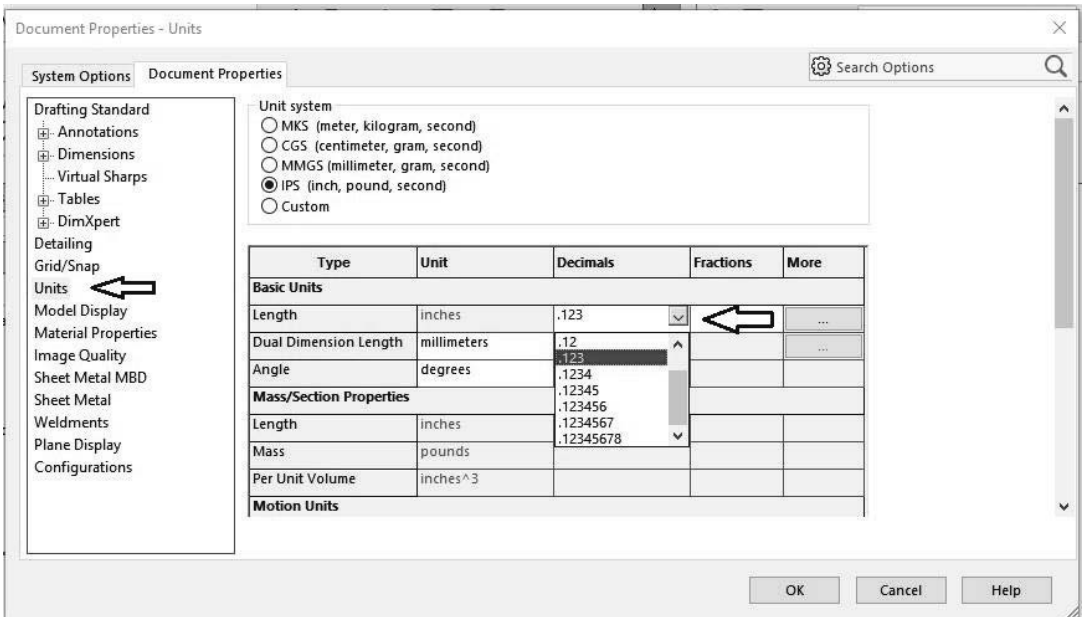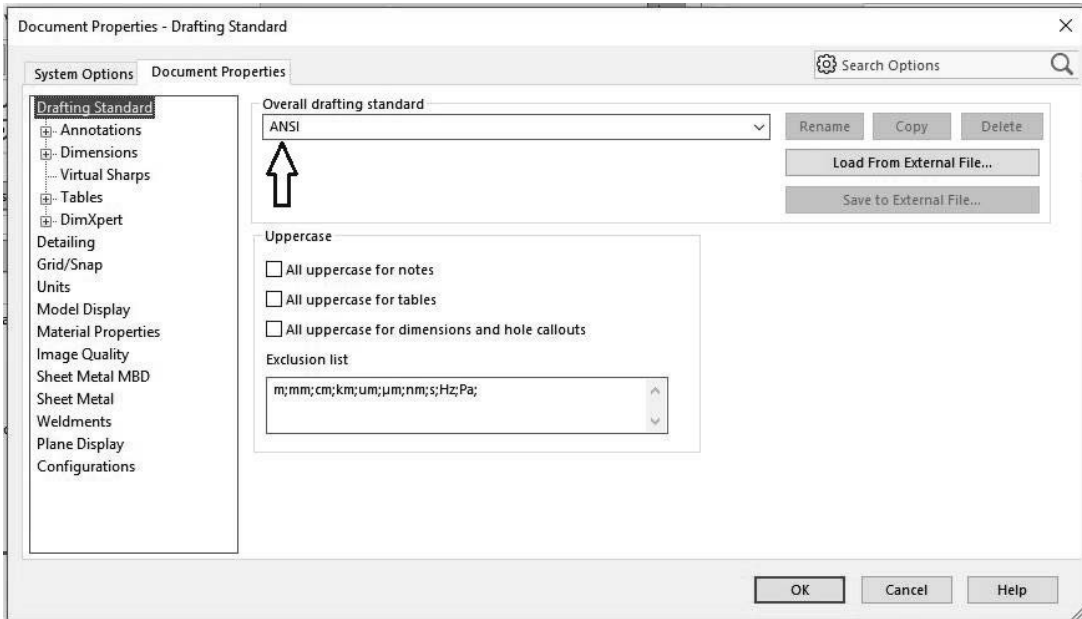
9. 💾 Under File at the top of the screen, pick **Save As…**
10. For the **Save As** type: select "**Part Template (*.prtdot)** from the pulldown menu. Enter "**Part_IPS_ANSI**" as the filename. At the top of the window navigate to the

SolidWorks Templates folder you created previously, and pick the **Save** button. If a file called "**Part.PRTDOT**" is in the same directory, you can delete it.
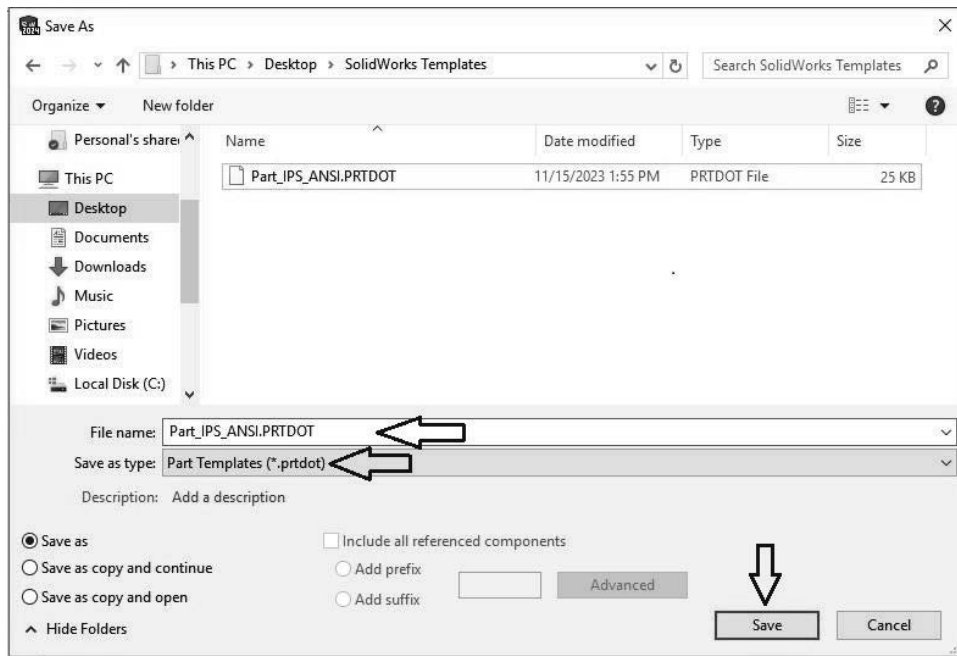

11. Under File, pick **Close**.

Now we need to add this folder to file locations that SOLIDWORKS can look into.

12. Under the Tools menu, pick **Options**. Select **File Locations** under System Options. Make sure **Document Templates** is selected as the **Show folders for:** option. Pick the **Add…** button to add the user-define template to the list of folders containing document templates. Locate the SolidWorks Templates folder on the Desktop, then pick the **Select Folder** button.
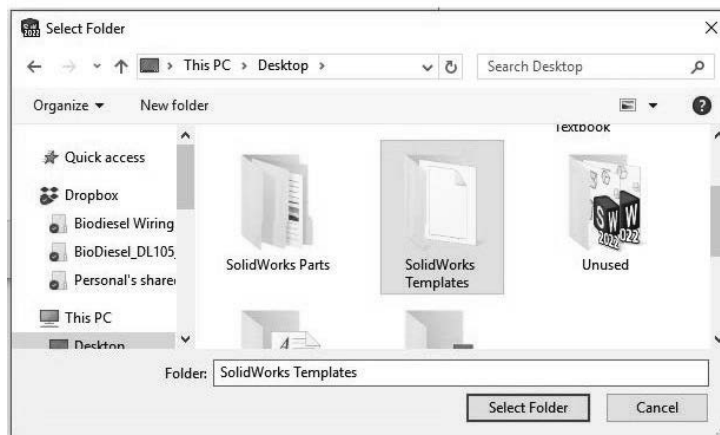

13. Pick the **OK** button. If a pop-up window appears with the question, "Would you like to make the following changes to your search paths?" Pick **Yes**.
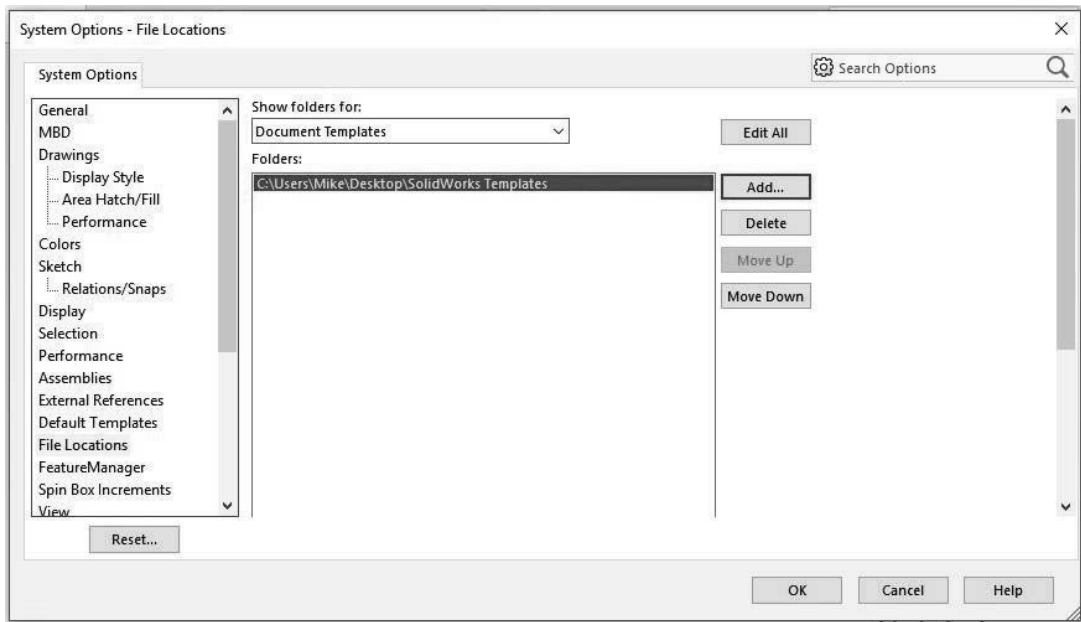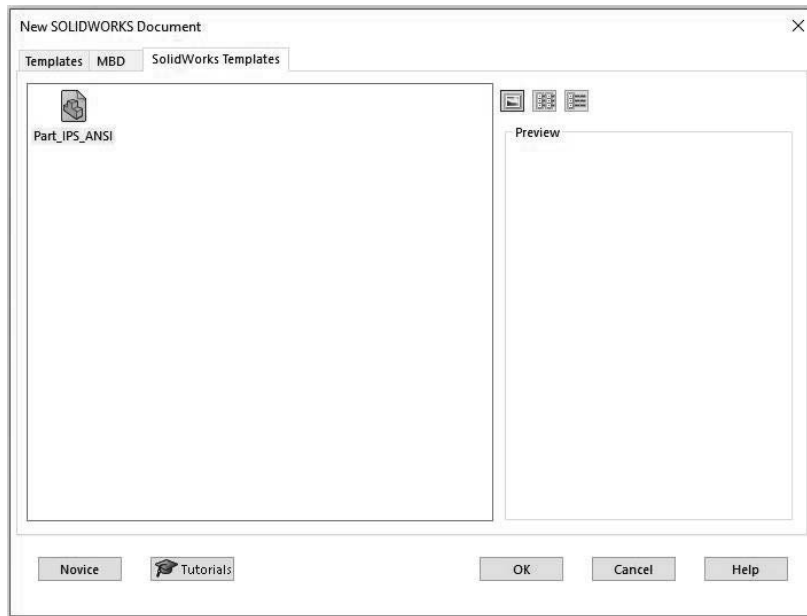
We will use this template for each new part we create so that we don't have to set these parameters each time we create a new part.

14. Under File, pick **New…** Pick the **Advanced** button in the lower-left corner of the window if SOLIDWORKS comes up in Novice mode.
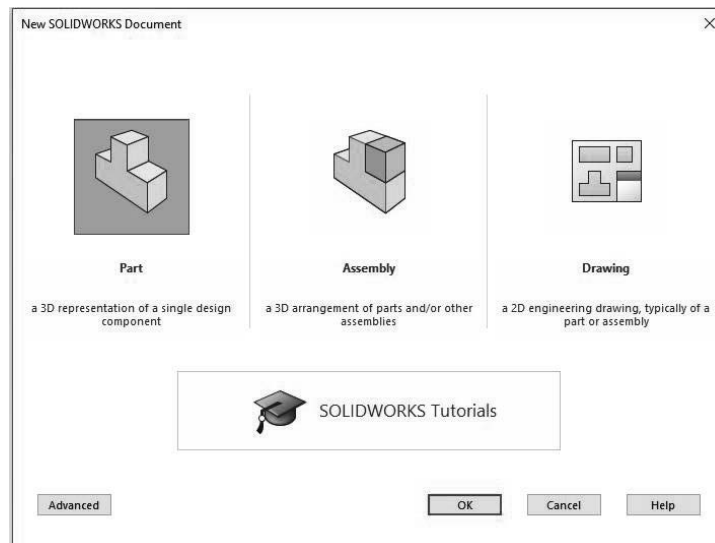

15. Under the SolidWorks Templates tab, pick the **Part_IPS_ANSI** icon. Pick **OK.**

We are back in SOLIDWORKS with all desired parameters already set. If the left **Extruded Boss/Base Toolbar** and the right **Sketching Toolbar** are not present, don't worry they will be added shortly.
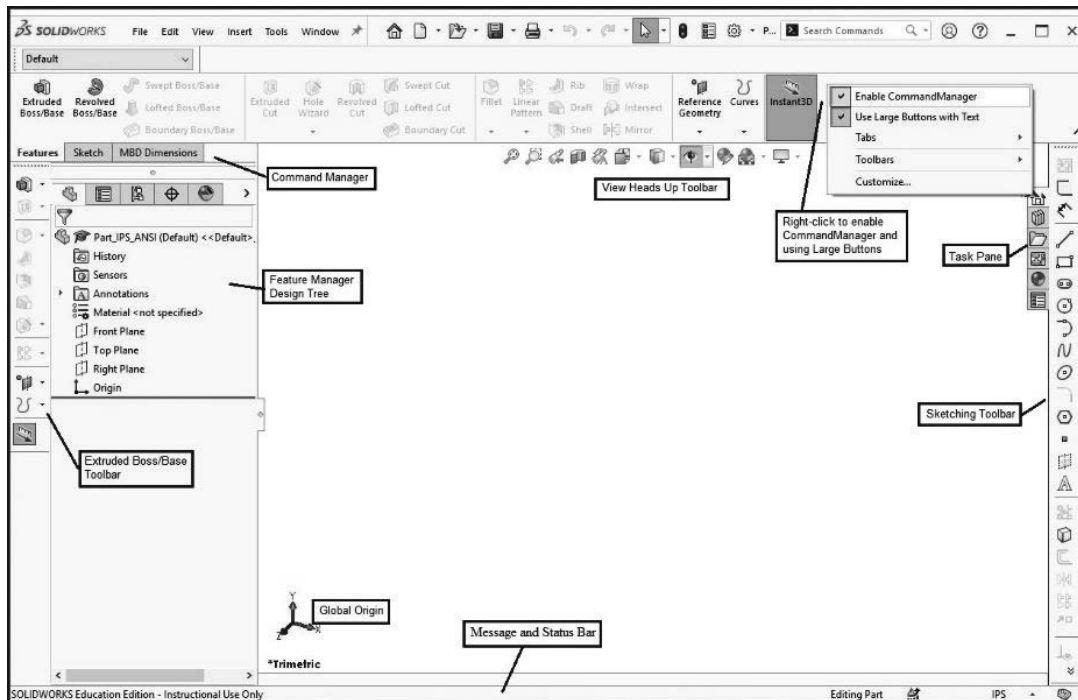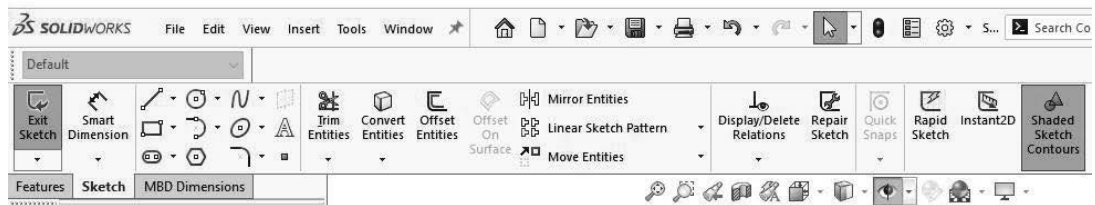

Note that I only have 3 tabs (Features, Sketch, and MBD Dimensions) listed in my working window because I have turned off the other tabs. You may see more tabs. Right-click at the top and select the **Tab** option to turn off the tabs you do not want to see or use.
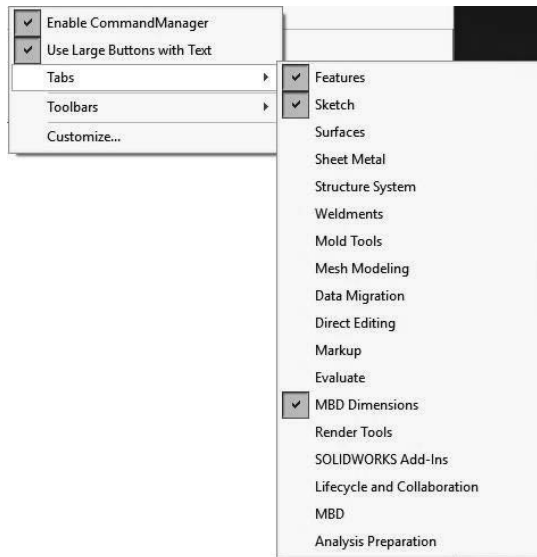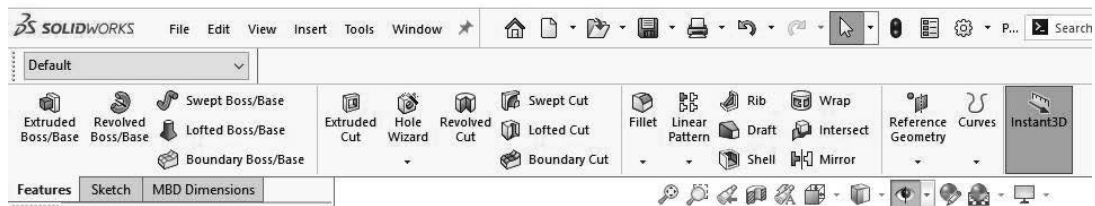
With **Use Large Buttons with Text** selected, the three tabs appear as follows.
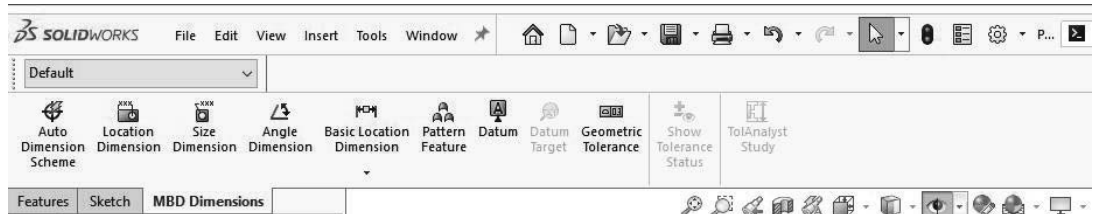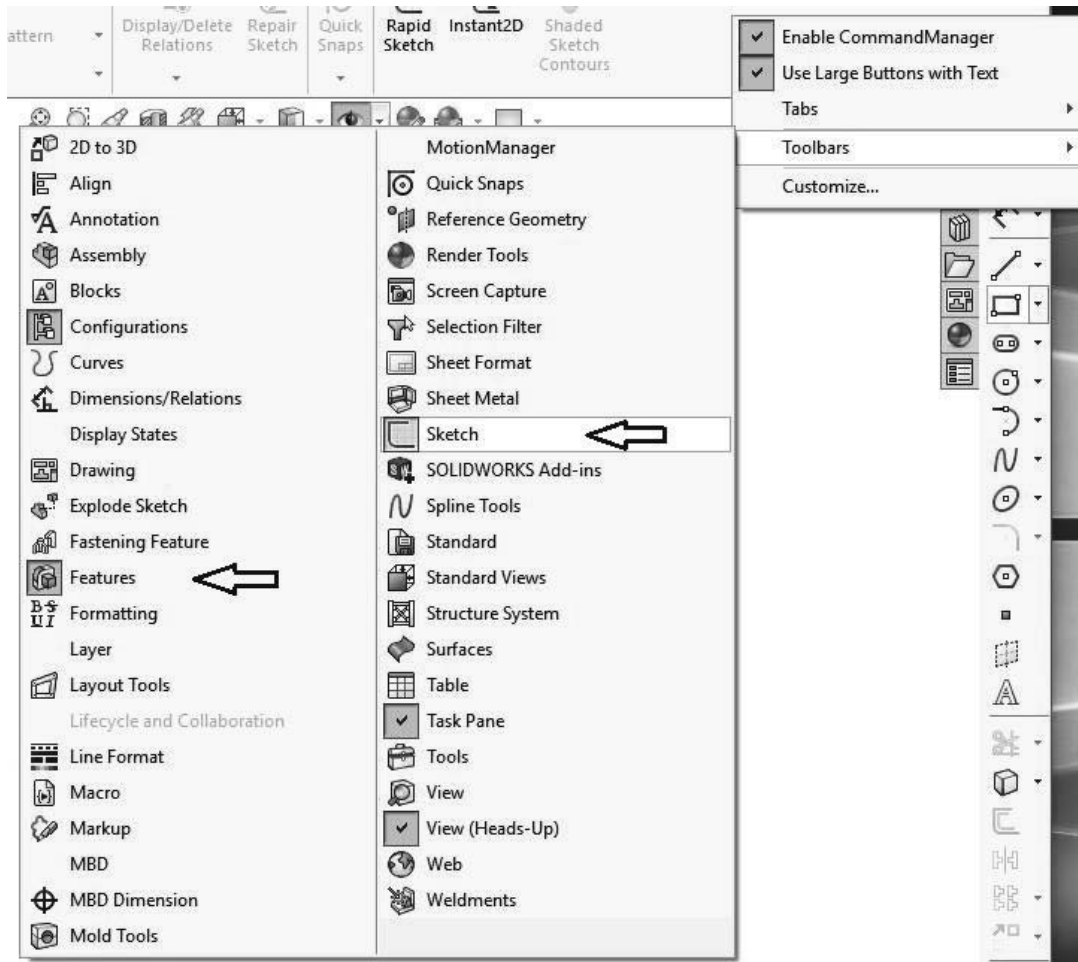
The Sketch tab.

The Features tab.

The MBD Dimensions tab.

You can also add the Sketch and Features tools along the side of the window. The **Sketch** tools will appear along the right side of the window. The **Feature** tools will appear along the left side of the window. These are added using **Toolbars** from the same pop-up menu.
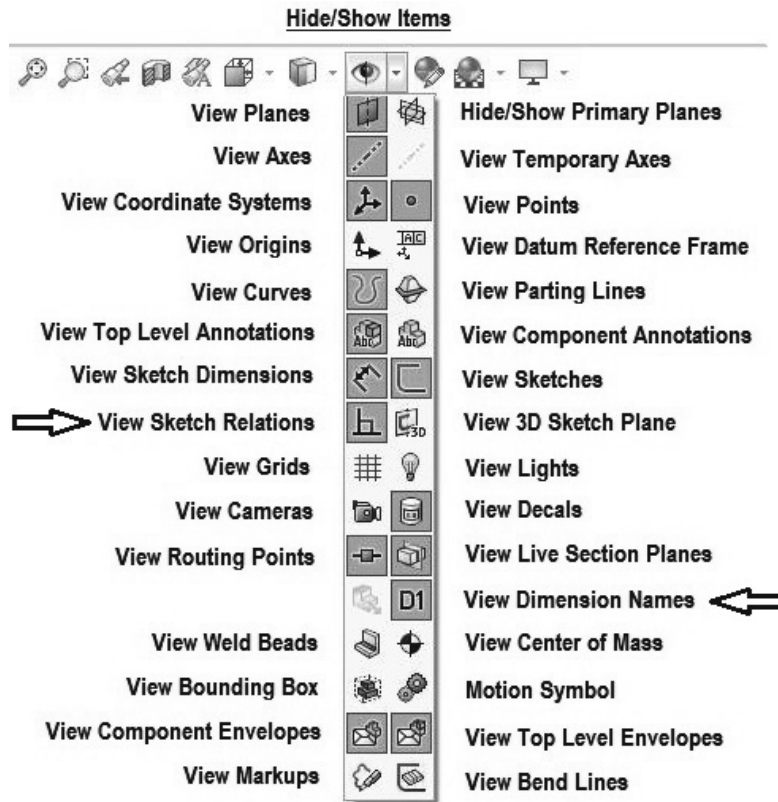
Selecting the **CommandManager** from the pop-up menu disables it. Picking it again enables it. Disabling the **Use Large Buttons with Text** option gives you more room to work so do this now.

16. [icon] To start a sketch, first select one of the three principal planes (Front Plane, Top Plane, or Right Plane) in the Feature Manager Design Tree, then pick the **Sketch** icon at the top of the window or along the right side of the window using the LMB. Pick the **Front Plane** for this exercise.

If you forget to pick the plane first, the three orthogonal planes are listed in the Feature Manager Design Tree so you can select the desired plane such as the **Front Plane**.

Before sketching it is a good idea to turn on **View Sketch Relations** so you can tell when a line is horizontal, vertical, parallel to another line, perpendicular to another line, tangent to a curve, etc.

17. [icon] Pick the **eyeball** icon in the **View Heads Up Toolbar**. [icon] Verify that the **View Sketch Relations** icon is highlighted, otherwise select it.

Hide/Show Items

View Planes		Hide/Show Primary Planes
View Axes		View Temporary Axes
View Coordinate Systems		View Points
View Origins		View Datum Reference Frame
View Curves		View Parting Lines
View Top Level Annotations		View Component Annotations
View Sketch Dimensions		View Sketches
⇒ View Sketch Relations		View 3D Sketch Plane
View Grids		View Lights
View Cameras		View Decals
View Routing Points		View Live Section Planes
	D1	View Dimension Names ⇐
View Weld Beads		View Center of Mass
View Bounding Box		Motion Symbol
View Component Envelopes		View Top Level Envelopes
View Markups		View Bend Lines

18. Select the **Line** tool, then draw a few lines similar to what is shown below. The exact locations or lengths of the lines don't matter. Typically, you want to start at the plane's origin. If you create a closed area, the area will be shaded as shown below and the line command will be paused for one mouse click. To stop a chain of connected lines, double-click the left mouse button (LMB). The line command is still active so you can continue to draw additional lines. To stop drawing lines, press the <ESC> key, or Right-click (RMB) and pick **Select** from the pop-up menu.
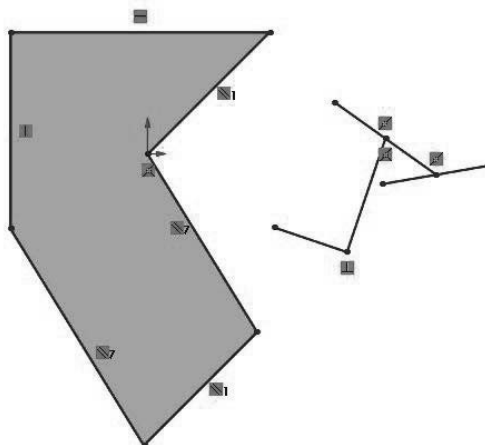
If the sketch relations do not show up on your sketch, see step 17.

In the figure above, the **Coincident** Sketch Relations icon indicates that the first point is coincident with the Front Plane's origin. There are two sets of parallel lines. There is a vertical and a horizontal line, and two lines that are perpendicular to each other. On the right, one line connects at the midpoint of another line and is coincident with that line. This information can be very helpful when sketching a given shape. A list of Sketch Relations is shown below.

Sketch Relations

|		
—	Horizontal	**Horizontal** forces the selected lines to be horizontal
|	Vertical	**Vertical** forces the selected lines to be vertical
╱	Collinear	**Collinear** for the selected lines to be in line with each other
⊥	Perpendicular	**Perpendicular** forces two lines to be perpendicular
╲	Parallel	**Parallel** forces the selected lines to be perpendicular
=	Equal	**Equal** force the selected objects to be equal in size
╱	Midpoint	**Midpoint** forces the selected object to connect at the midpoint
人	Coincident	**Coincident** forces two points or objects to be connected
⚝	Fix	**Fix** forces the selected object to be fixed in the window
⌀	Tangent	**Tangent** forces a curve to be tangent to a curve or line

If your sketched lines do not reflect the desired Sketch Relations, you can select the appropriate lines by picking the first line, then holding down the <CTRL> key and picking the second line. A list of possible sketch relations will appear on the left side of the window. Pick the appropriate sketch relation from the list.

19. Move to the upper portion of the sketching window and hold the LMB down. Use a sweeping motion to surround all of the lines previously drawn. Release the LMB, then press the **<Delete>** key to delete all of the selected lines.

20. ⊡ Select the **Circle** icon from the sketch toolbar. Pick a point in the window, then move the mouse to a new location. A Circle's center will be located at the first point selected and the size will be dictated by the second mouse pick. Draw several circles similar to what is shown below.

21. Pick the center of the circle at the origin, hold down the <Ctrl> key, and pick the center of the smaller circle below it. In the Add Relations window, pick the Vertical icon. Pick the green checkmark.

22. Pick the center of the left circle, then hold down the <Ctrl> key, and pick the center of the right circle. In the Add Relations window, pick the Horizontal icon. Pick the green checkmark.

23. Pick the edge of the left circle, then hold down the <Ctrl> key, and pick the edge of the right circle. In the Add Relations window, pick the Equal icon. Pick the green checkmark.

24. Pick the edge of the left circle, then hold down the <Ctrl> key, and pick the edge of the circle centered at the origin. In the Add Relations window, pick the Tangent icon. Pick the green checkmark.
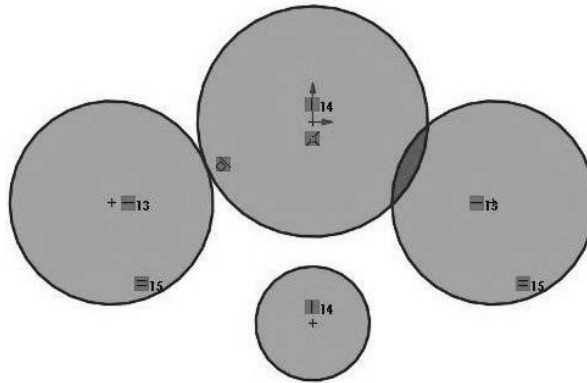
The first circle was drawn at the origin of the sketch window. The smaller circle is directly below the first circle as indicated by the Vertical relation. The left and right circles are equal in size and their centers are along the same horizontal axes.

25. Select the **Line** tool again, then add four lines that are tangent to the four circles as shown. If any of the lines are not tangent to the given circle, select the circle near the endpoint of the line, and pick Tangent from the Sketch Relations window on the left. If another relation is present other than the coincident relation at the end of a given line, pick the relation with the LMB, then press the <Delete> key to remove it. Each line segment should have a Tangent and Coincident relation attached. See the figure below.
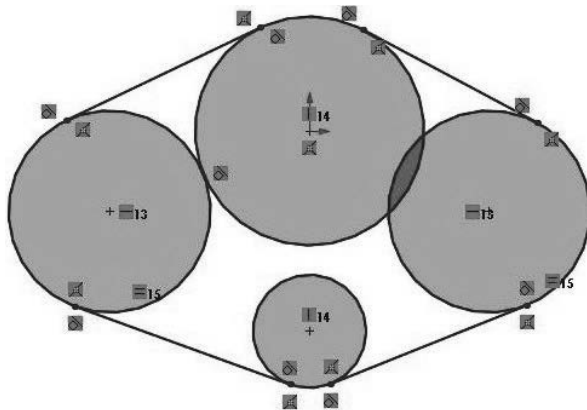

Use the **Trim Entities** tool to remove unwanted portions of your sketch.

26. Select the **Trim Entities** icon. Move the mouse inside the four tangent lines, then press down and hold the LMB, and drag the mouse cursor around without touching any of the four lines or the outside region of the four circles. Your sketch should look similar to the figure below. If you accidentally touch a line or the outer portion of a circle, undo the changes by pressing <Ctrl>+Z and try again.
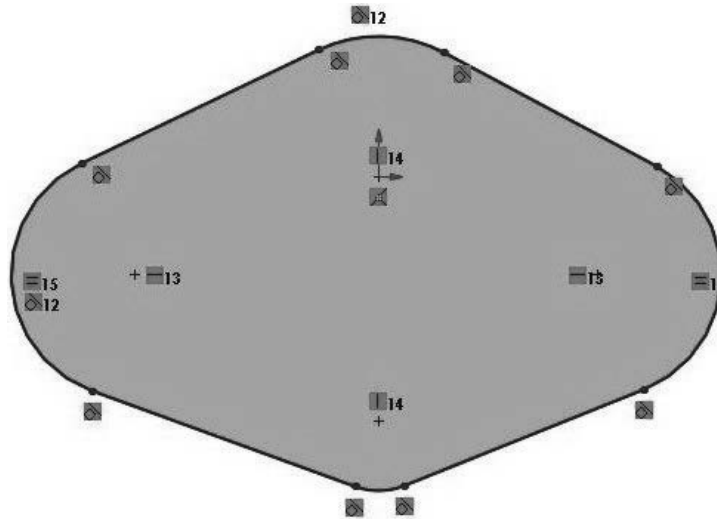
At the bottom of the window, the phrase Under Defined is visible. This is because we haven't sized any of the circles or located the centers of any of the circles. We will do this now. Also, note that the sketch is not currently symmetrical.

27. Pick the **Smart Dimension** icon. Pick the left circle, then move away from it and pick the location for the size dimension. Type 1.5 for the value, then press the <Enter> key or pick the green checkmark in the pop-up window.
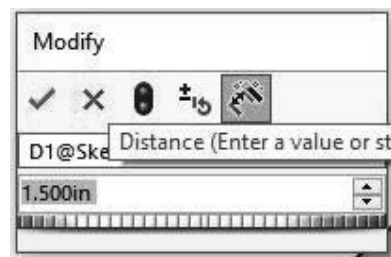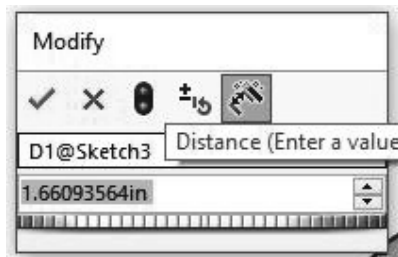
The radius of the left arc will be resized to 1.50 inches. Note that the right arc will also be 1.50 inches since the two arcs were set equal in size previously. Note that the dimension names will show up as well if the **View Dimension Names** is turned on.

28. Pick the top arc and set its size to 1.75 inches.
29. Pick the bottom arc and set its size to 1.00 inches.
30. Pick the center of the left arc, then the center of the right arc. Move the mouse cursor down and note that a dimension appears between the centers of the two arcs. Set its value to 5.50 inches.
31. Set the distance between the top arc's center and the bottom arc's center equal to 3.00 inches. Dimension the distance between the top arc's center and the left or right arc's center. It should be a driven dimension equal to 1.73 inches.

At the bottom of the window, the phrase Under Defined is still showing so there is at least one more dimension or sketch relation missing. Let's make the object symmetrical. To do this, we need to add a centerline, and then add a symmetric relation.

32. Pick the down arrow to the right of the Line icon to bring up a pop-up menu. Pick **Centerline** from this menu, then draw a vertical centerline through the origin.

/	Line
✓	Centerline
↘	Midpoint Line

33. Pick the center of the left arc. Hold down the <Ctrl> key and pick the center of the right arc, then pick the centerline. Select **Symmetric** from the Add Relation window. (You can pick these three entities in any order.)

At the bottom of the window, the phrase **Fully Defined** appears, and your sketch should look exactly like the sketch below if the **View Dimension Names** option in the **Hide/Show Items** feature is turned on. (See page 1-11.)
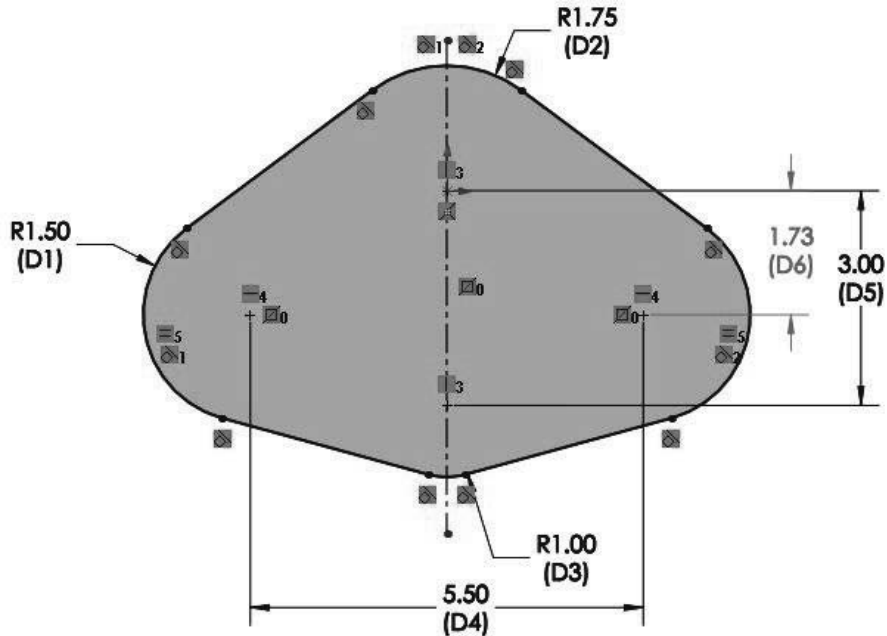

Let's exit from the sketch, and extrude this shape to make our first 3D part with a depth of 2.00 inches.

34. Pick the shaded **Sketch** icon on the right or the **Exit Sketch** icon at the top of the screen if the **Use Large Buttons with Text** feature is enabled.

35. With the sketch highlighted in the Feature Manager Design Tree, pick the **Extruded Boss/Base** icon. Leave the direction as Blind. Set the depth in Direction 1 to 2.00 inches. Pick the green checkmark to complete the extrusion.
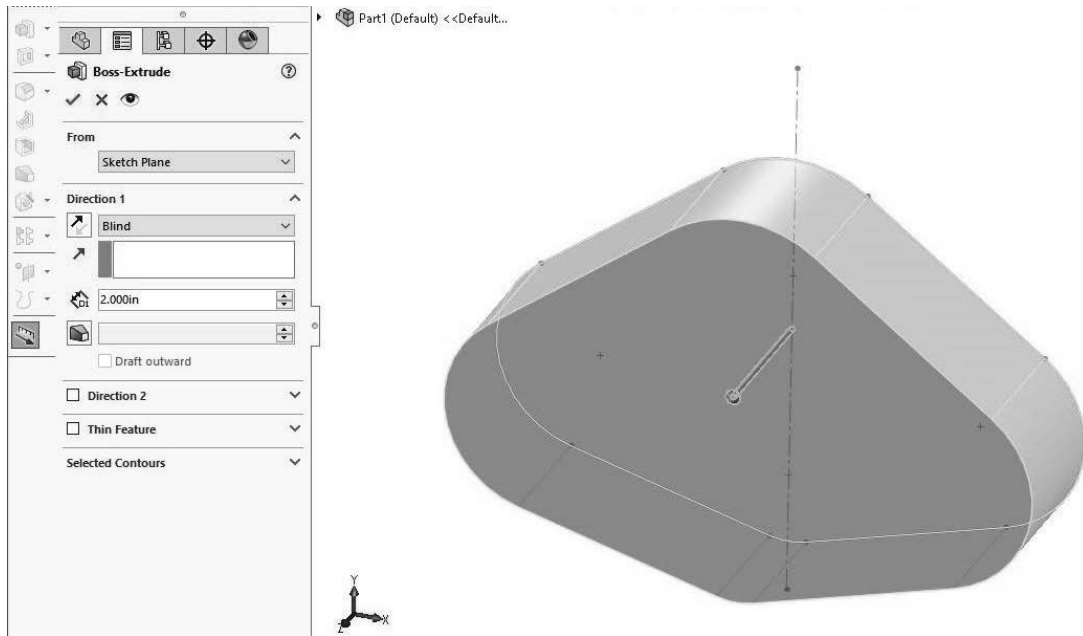
The final extruded part appears on the screen.
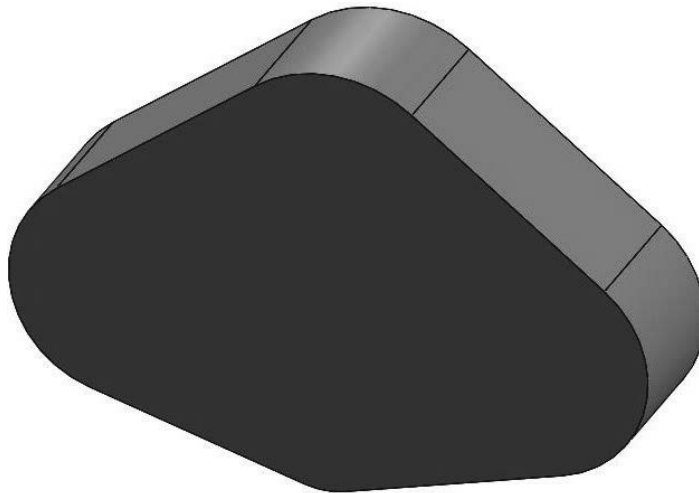
36. If necessary, press the <F> key to fit the part on the screen.
37. Right-click on the **Material** icon in the **FeatureManager Design Tree** and select **Plain Carbon Steel** using the LMB. Notice that the selected material now appears in the **FeatureManager Design Tree**.
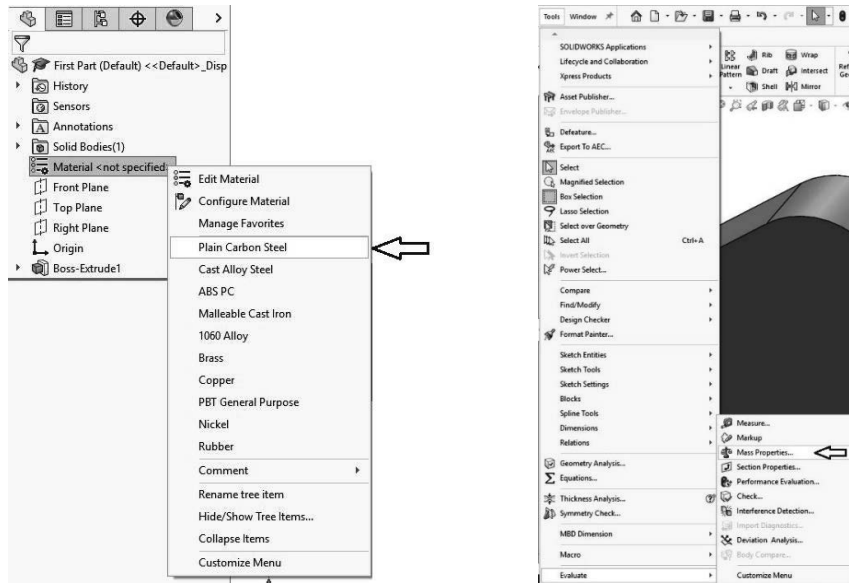
38. Under the File menu, pick **Save**. Name the part as "**First Part**". Make sure you are saving the part file in the SolidWorks Parts folder you created previously. Pick **Save**.

39. Under **Tools**, select **Evaluate**, then select **Mass properties...** The following window appears. The mass should be 18.88 pounds and the volume should be 67.01 cubic inches if you properly dimensioned the part and added the appropriate sketch relations. If not, go back and check your construction procedure.
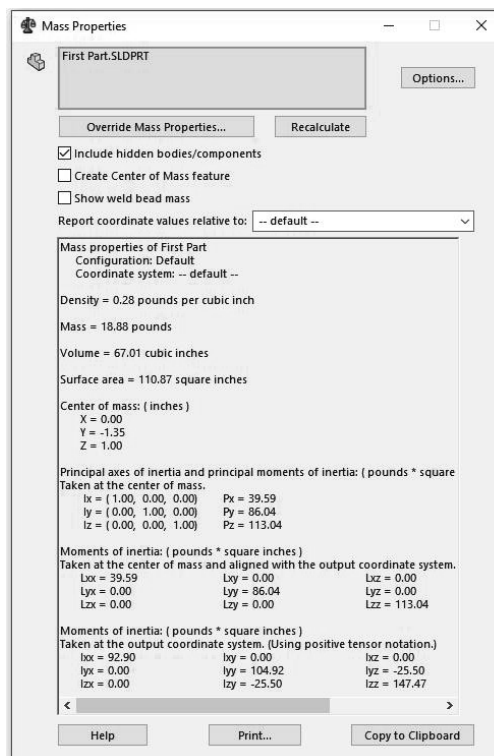
40. Pick the X in the upper right corner of the window to close the Mass Properties window.

41. If your mass or volume is different, right-click on **Boss-Extrude1** and select **Edit Sketch**, then check over your sketch for this part. Maybe you are missing a sketch relation. After correcting your sketch, exit your sketch, and check **Mass properties...** again.
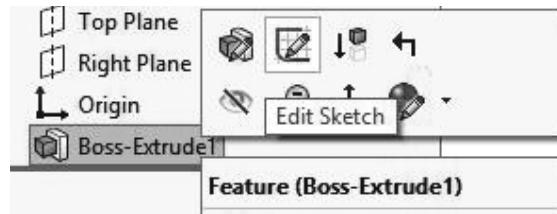
42. Pick **Save**.
43. Under the File menu again, pick **Close**.
44. Under the File menu, pick **Exit** to exit from SOLIDWORKS.