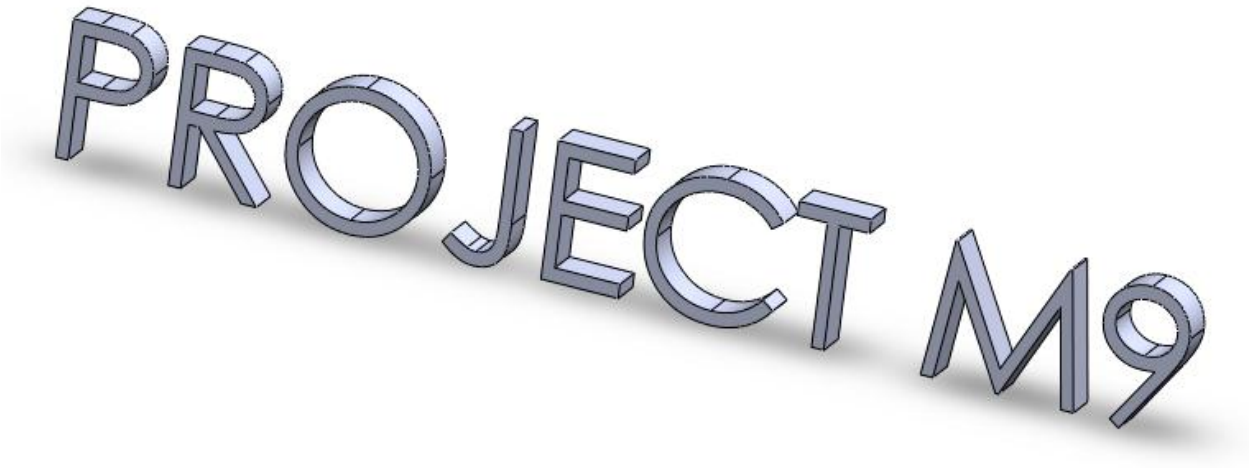
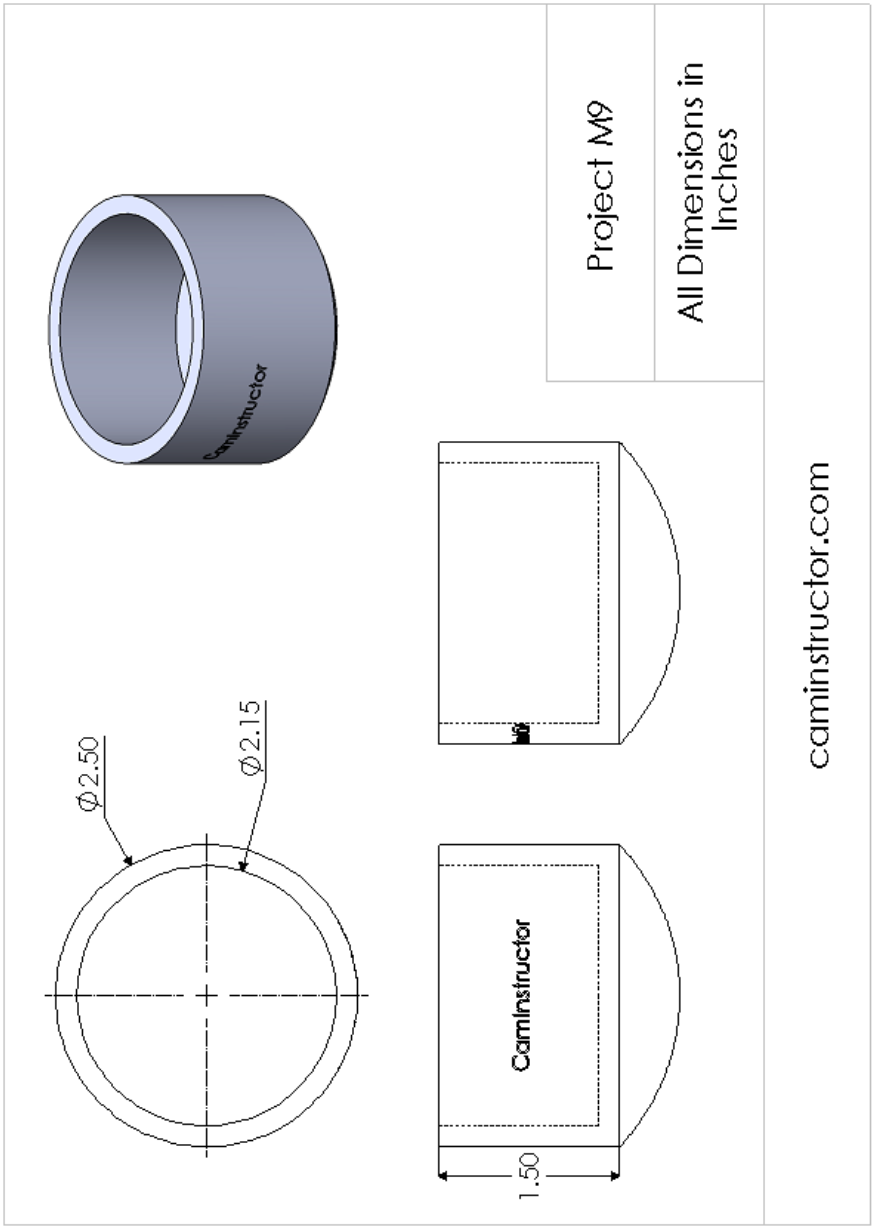


camInstructor

Figure 9.: Project M 9

In this Project you will complete the solid model using the various solid commands in SolidWorks.

- Extruded Boss
- Shell
- Dome
- Wrap


Project M9
All Dimensions in Inches
caminstructor.com
CamInstructor
Ø2.50
Ø2.15
1.50

1. Create a new Sketch

To create a **new** SolidWorks document please refer to the **Getting Started** chapter.

Note: *For this Project you will choose the* ***Top Plane*** *in procedure* ***6.2****. of the* ***Getting Started*** *chapter.*

2. Create the Extruded Boss

2.1. Draw the Sketch

- **2.1.1.** Left click on the mouse **once** on the **Circle** button in the **Sketch CommandManager**.

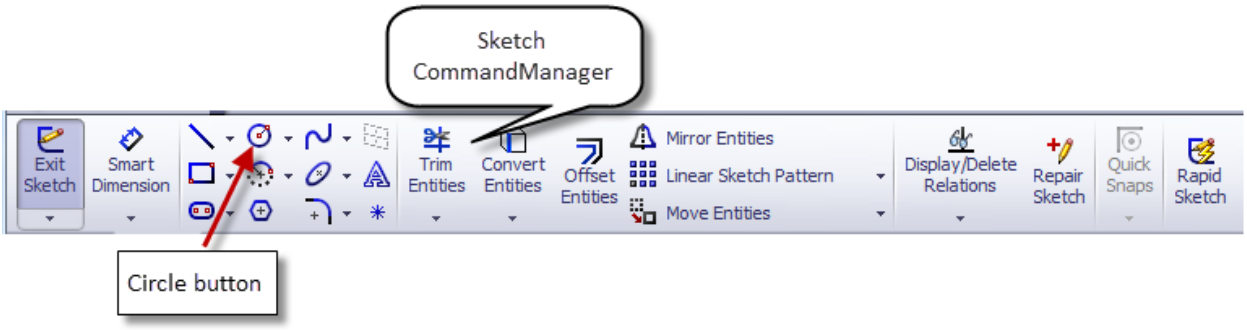


Figure 9.11.: Circle button in the Sketch CommandManager

- **2.1.2.** Move the cursor to the **SolidWorks drawing area** and left click on the mouse **once** on the **Origin** (the **Origin** is represented by an **orange** circle on the intersection of the 2 red lines with arrows).

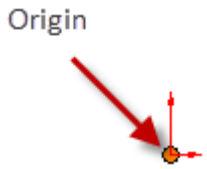


Figure 9.12.: Origin

- **2.1.3.** Move the cursor **upward** to a radius of approximately **1.25** (between **1** and **1.25**) and left click on the mouse **once**

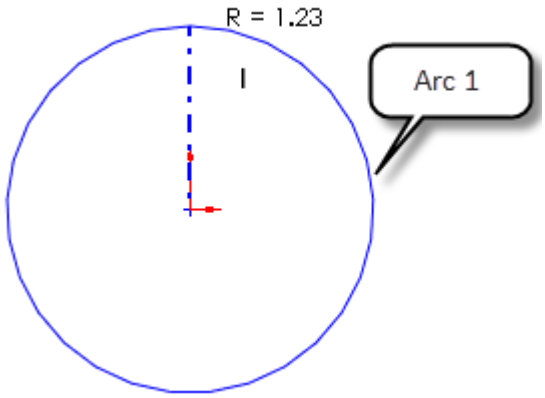


Figure 9.13.: Arc 1

- **2.1.4.** Press the **ESC** key on the keyboard to **exit** the **Circle PropertyManager**.

2.2. Dimension the Sketch

- **2.2.1.** Left click on the mouse **once** on the **Smart Dimension** button in the **Sketch CommandManager**.

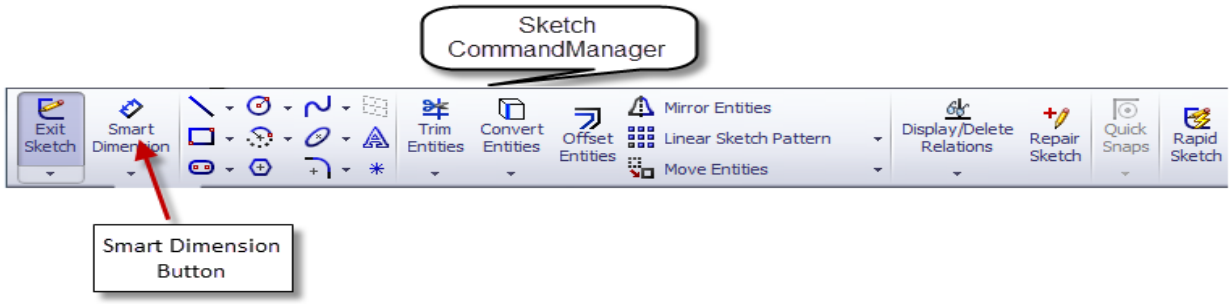


Figure 9.14.: Smart Dimension in the Sketch CommandManager

- **2.2.2.** Move the cursor to the **SolidWorks drawing area** and left click on the mouse **once** on **Arc 1**.

- **2.2.3.** Position the cursor away from **Arc 1** and left click on the mouse **once**.

- **2.2.4.** A **Modify** dialog box appears. Type a value of **2.5** .

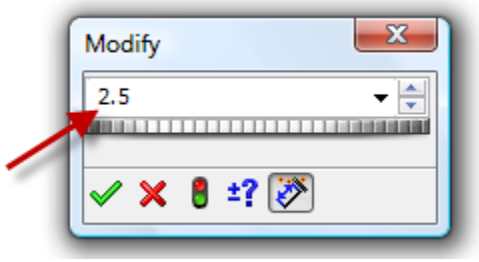


Figure 9.15.: Modify dialog box

- **2.2.5.** Press the **Enter** key on the keyboard. You will see the **updated** dimension displayed.
- **2.2.6.** Press the **Esc** key on the keyboard to **exit** the **Dimension Value PropertyManager.**

Note: When you have done dimensioning the sketch entities, the sketch will be **Fully Defined**. The sketch entities will be shown in **black**. ***Fully Defined*** *is shown on the* ***status bar*** *of SolidWorks.*

Figure 9.16.: Fully Defined on the status bar

2.3. Extrude the Sketch

- **2.3.1.** Left click on the mouse **once** on the **Features** tab to display the **Features CommandManager**.

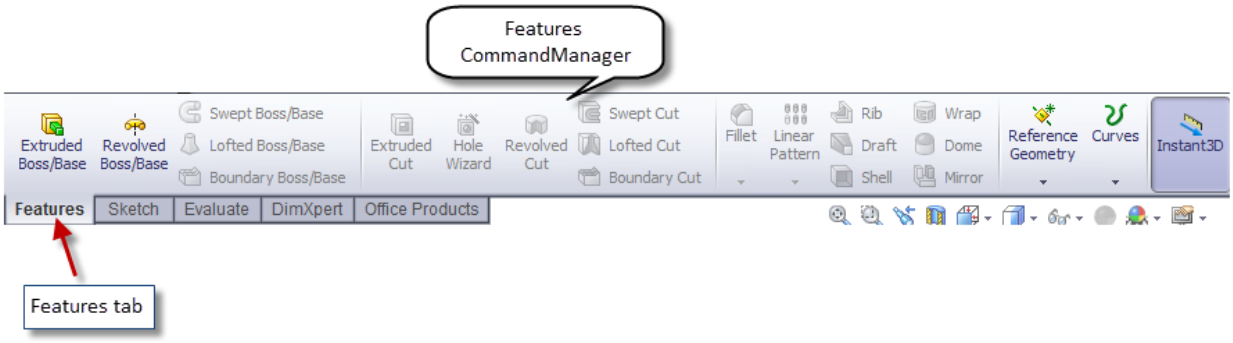


Figure 9.17.: Features tab to display the Features CommandManager

- **2.3.2.** Left click on the mouse **once** on the **Extruded Boss/Base** button in the **Features CommandManager**.

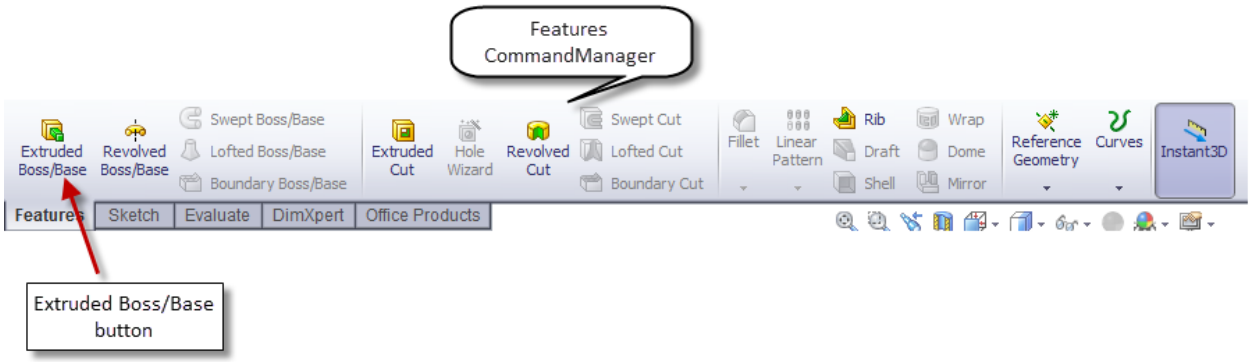


Figure 9.18.: Extruded Boss/Base button from the Features CommandManager

*The **Exrude PropertyManager** will appear. The **Extrude PropertyManager** allows you to define and edit the characteristics of the extrude sketch.*

***Note:** The sketch will orient to the **Trimetric view** as shown on the bottom left screen of the **SolidWorks drawing area**.*

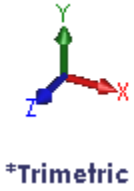


Figure 9.19.: Trimetric

- **2.3.3.** In the **Extrude PropertyManager** under **Direction 1** make sure the **End Condition** field is set to Midplane.

Note: *The **End Condition** set to **Mid Plane** allows you to set the extrude depth amount equally on both sides of the selected sketch.*

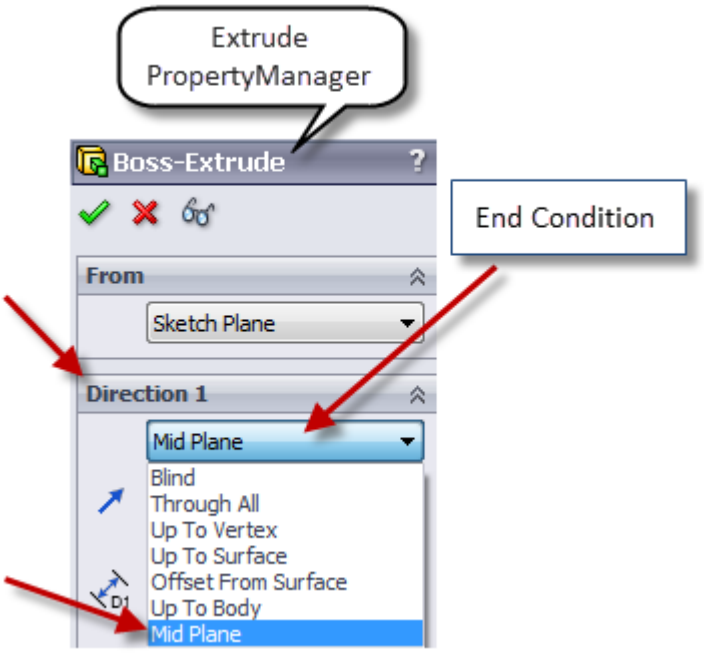


Figure 9.2.: Mid Plane in the Extrude PropertyManager

- **2.3.4.** In the **Extrude PropertyManager** type a value of **1.5** in the **Depth** field and press the **Enter** key on the keyboard.

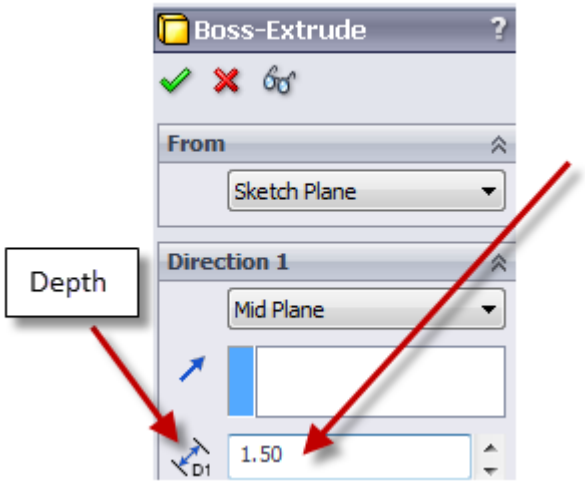


Figure 9.21.: Depth field in the Extrude PropertyManager

In the **SolidWorks drawing area** you will see the preview of the solid based on the parameters in the **Extrude PropertyManager.**

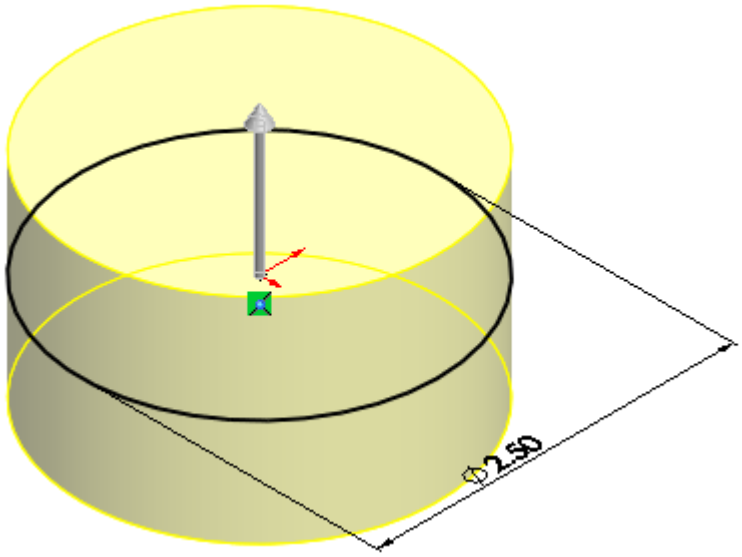


Figure 9.22.: Preview of the extruded solid

- **2.3.5**. In the **Extrude PropertyManager** click on **OK (green check mark)** to **exit**.

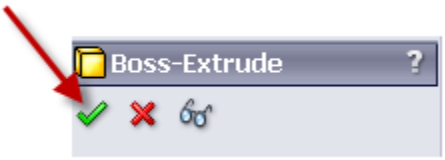


Figure 9.23.: Ok in the Extrude PropertyManager

3. Create the Shell Features

3.1. Left click on the mouse **once** on the **Shell** button in the **Features CommandManager**.

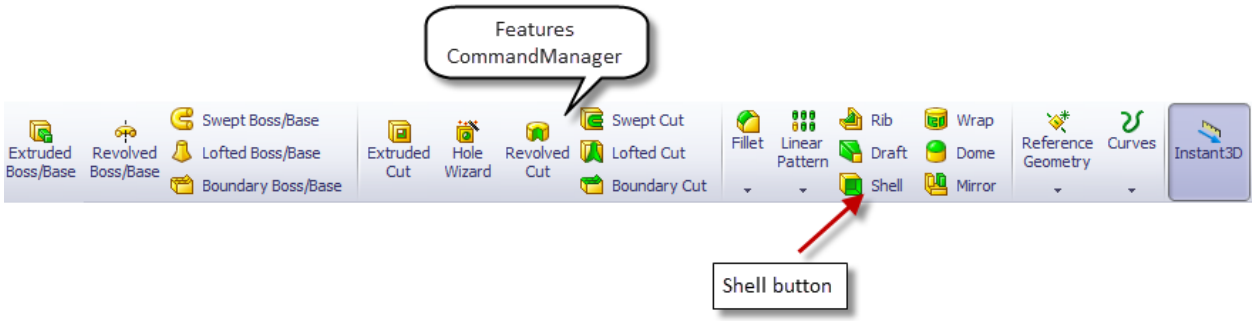


Figure 9.24.: Shell button in the Features CommandManager

*The **Shell Features PropertyManager** will appear. The **Shell Features PropertyManager** allows you to define and edit the characteristics of the hollow feature created.*

3.2. In the **Shell Features PropertyManager** type a value of **0.175** in the **Thickness** field and press the **Enter** key on the keyboard.

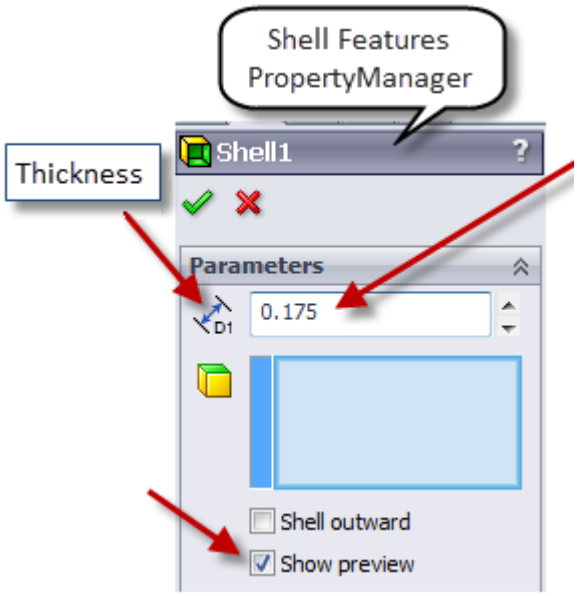


Figure 9.25.: Depth field in the Shell Features PropetyManager

***Note:** Check the box besides **Show preview**.*

***Note:** The **Thickness** field represents the wall thickness.*

3.3. Move the cursor to the **SolidWorks drawing area** and left click on the mouse **once** on the **top face** of the extruded solid as shown below .

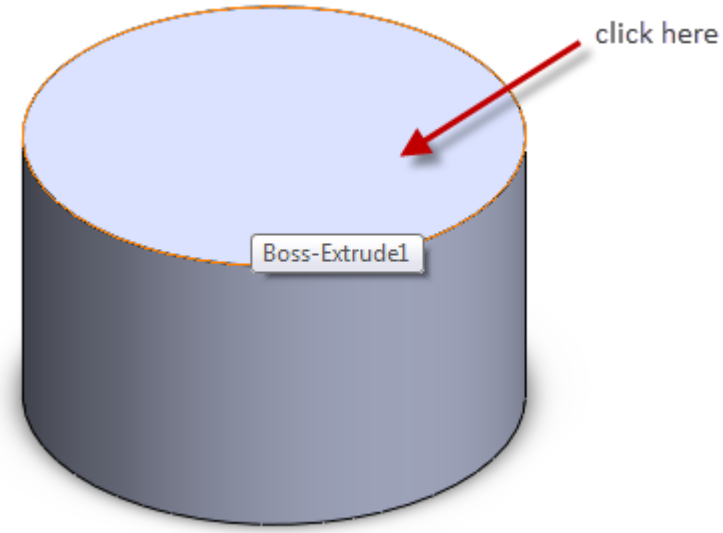


Figure 9.26.: Top face

Note: *In the* ***Shell Features*** **PropertyManager Face** ***<1>*** *will now be displayed in the box besides* ***Faces to Remove****.*

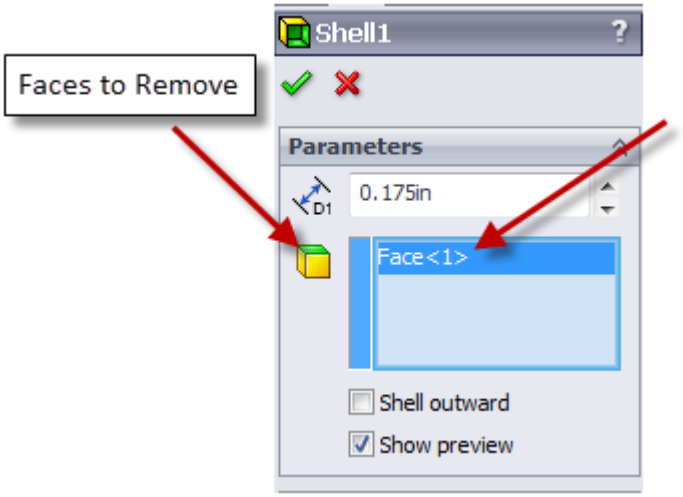


Figure 9.27.: Faces to Remove in the Shell Features PropertyManager

3.4. In the **Shell Features PropertyManager** click on **OK (green check mark)** to **exit**.

Figure 9.28.: OK in the Shell Features PropertyManager

In the **SolidWorks drawing area** the solid will be shown as below.

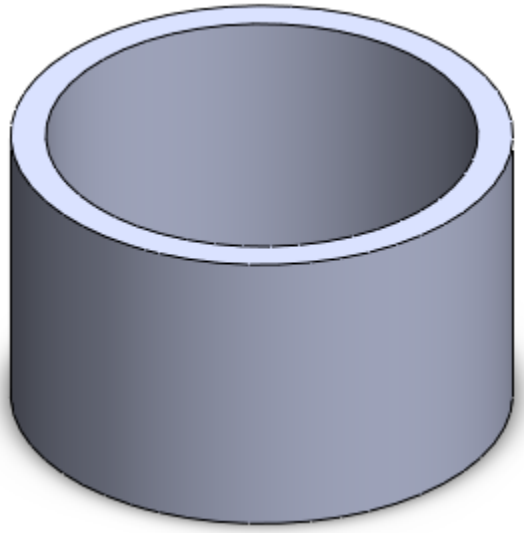


Figure 9.29.: Solid

4. Create the Dome Features

4.1. Left click on the mouse **once** on the **Dome** button in the **Features CommandManager**.

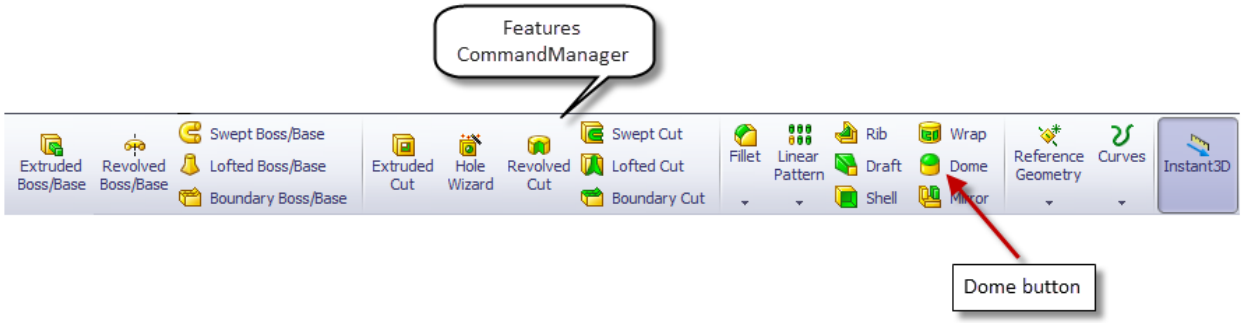


Figure 9.3.: Dome button in the Features CommandManager

*The **Dome PropertyManager** will appear. The **Dome PropertyManager** allows you to define the characteristics of the dome features. The dome features is based on cylindrical, conical, and polygon shapes*.

4.2. In the **Dome PropertyManager** under **Parameters** type a value of **0.5** in the **Distance** field and press the **Enter** key on the keyboard.

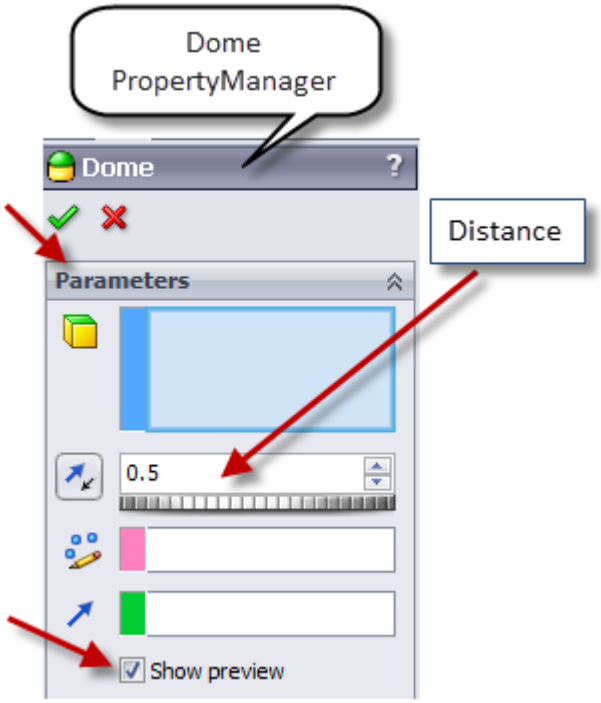


Figure 9.31.: Distance field in the Dome PropertyManager

***Note:** Check the box besides **Show preview**.*

4.3. Move the cursor to the **SolidWorks drawing area** and left click on the mouse **once** on the **bottom face** of the extruded solid as shown below.

Note: *With the middle mouse button held rotate the part to select the bottom face.*

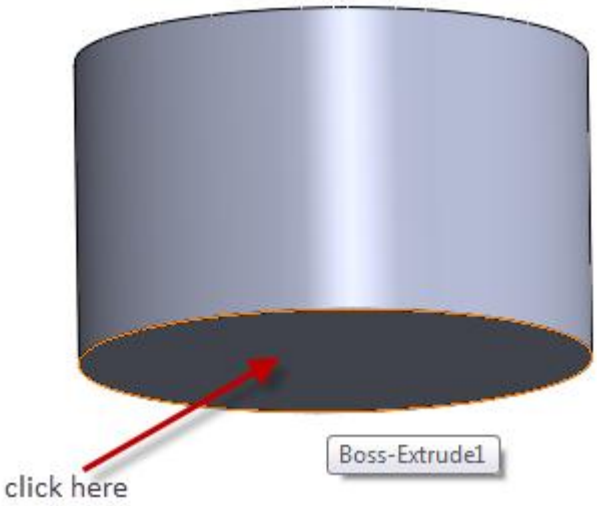


Figure 9.32.: bottom of the extruded solid

Note: *In the **Dome PropertyManager**, **Face <1>** will be displayed in the box besides the **Faces to Dome** field.*

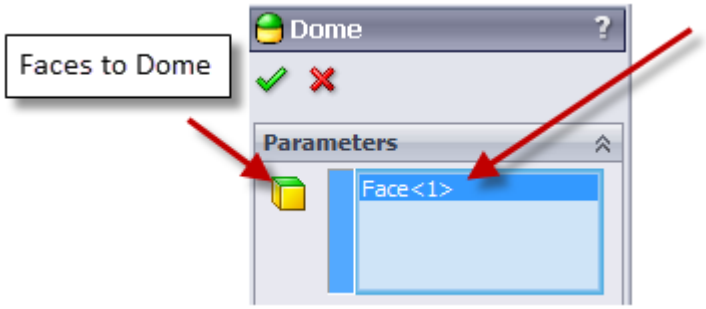


Figure 9.33.: Faces to Dome in the Dome PropertyManager

4.4. In the **Dome PropertyManager** click on **OK (green check mark)** to **exit**.

Figure 9.34.: OK in the Dome PropertyManager

In the **SolidWorks drawing area**, the solid will look as shown below.

Figure 9.35.: Solid

5. Create the Wrap Features

5.1. Draw the Sketch

- **5.1.1.** Move the cursor to the **FeatureManager design tree** and right click on the mouse **once** on **Front Plane**.

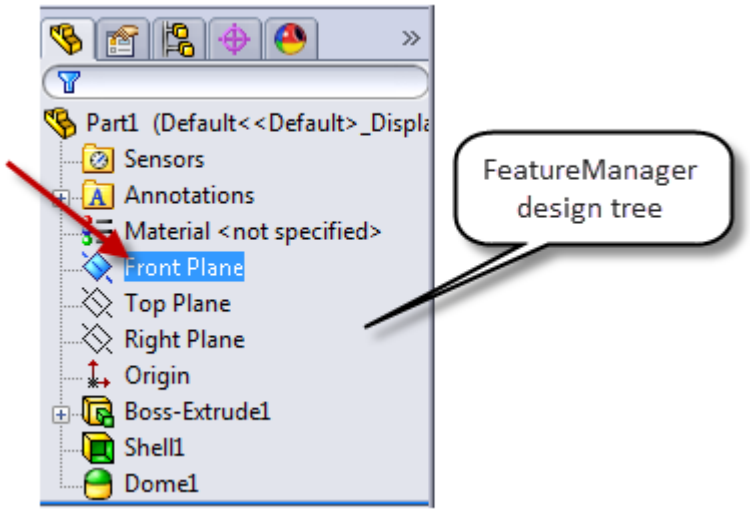


Figure 9.36.: Front Plane in the FeatureManager design tree

Note: *In the* ***FeatureManager design tree Front Plane*** *will highlight in* ***blue****.*

- **5.1.2.** In the **Pop-Up** toolbar left click on the mouse **once** on the **Sketch** icon.

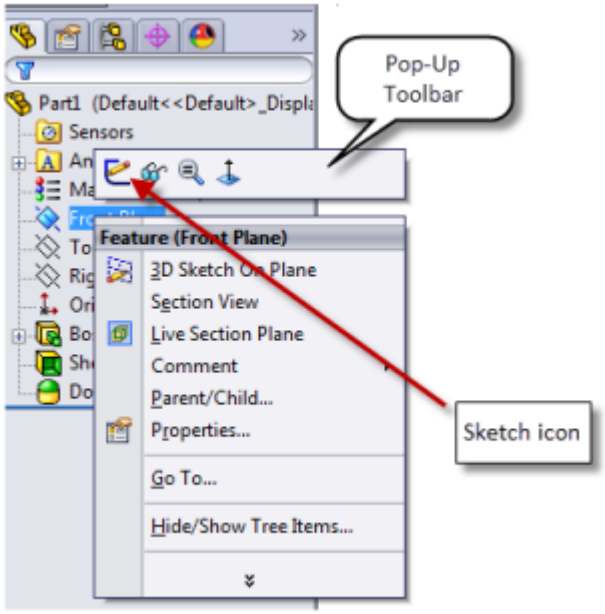


Figure 9.37.: Sketch Icon in the Pop-Up Toolbar

- **5.1.3.** Press the **space bar** on the keyboard **once**.

*The **Orientation** window will appear. The **Orientation** window allows you to orient the entities in the **SolidWorks Drawing area** to the view you choose.*

- **5.1.4.** In the **Orientation** left click on the mouse **twice** on***Normal To**.

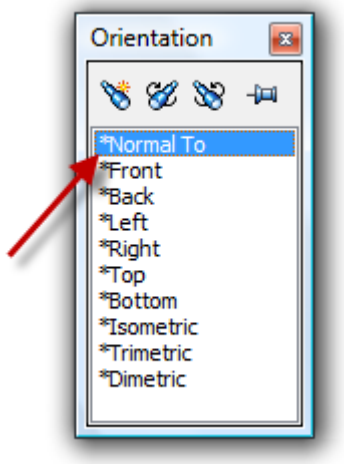


Figure 9.38.: *Normal To in the Orientation window

- **5.1.5.** In the **Sketch CommandManager** left click on the mouse **once** on the **Flyout** beside the **Line** button and select the **Centerline** button.

Figure 9.39.: Centerline button in the Sketch CommandManager

- **5.1.6.** Move the cursor to the **SolidWorks drawing area** and left click on the mouse **once** on the **Midpoint** of the left edge of the extruded solid and then **once** on the **Midpoint** of the right edge of the extruded solid as shown below.

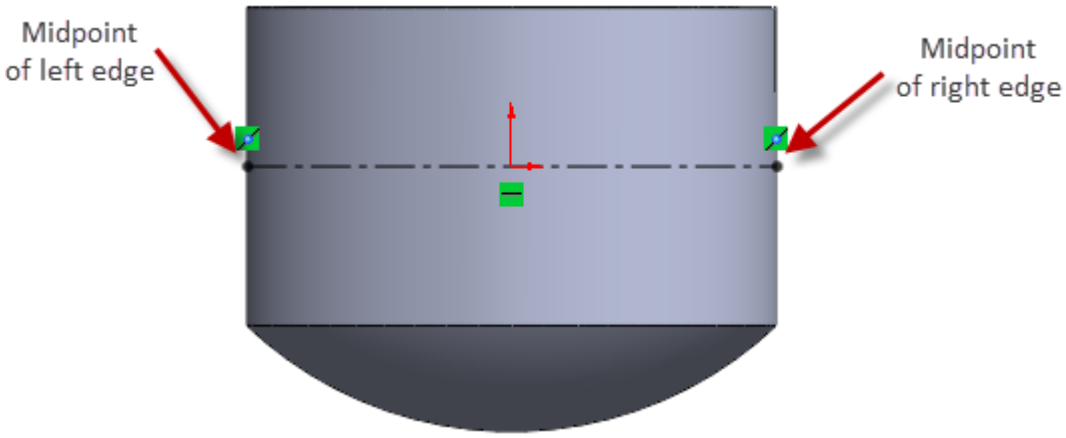


Figure 9.4.: Midpoint of the edges of the extruded solid

- **5.1.7.** Press the **Esc** key on the keyboard to **exit** the **Line Properties PropertyManager**.

- **5.1.8.** Left click on the mouse **once** on the **Text** button in the **Sketch CommandManager**.

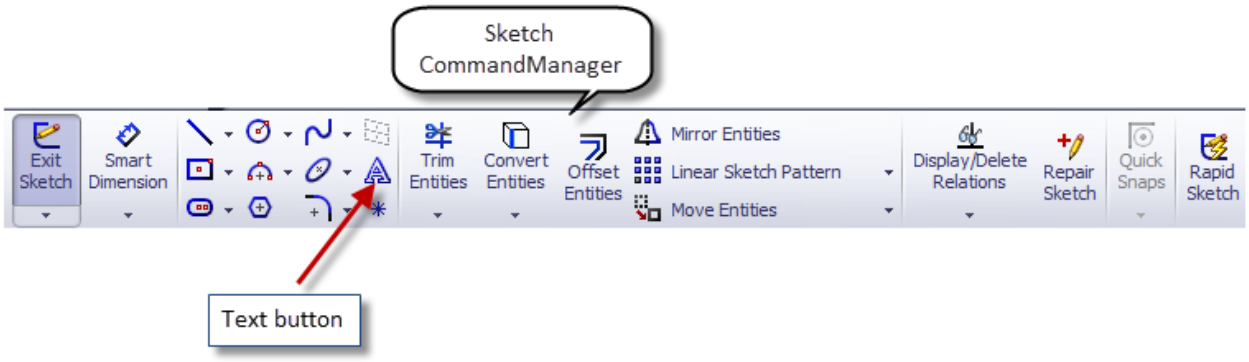


Figure 9.41.: Text button in the Sketch CommandManager

*The **Sketch Text PropertyManager** will appear. The **Sketch Text PropertyManager** allows you to sketch a text on a face of a solid.*

- **5.1.9.** Move the cursor to the **SolidWorks drawing area** and left click on the mouse **once** on **Line 1**.

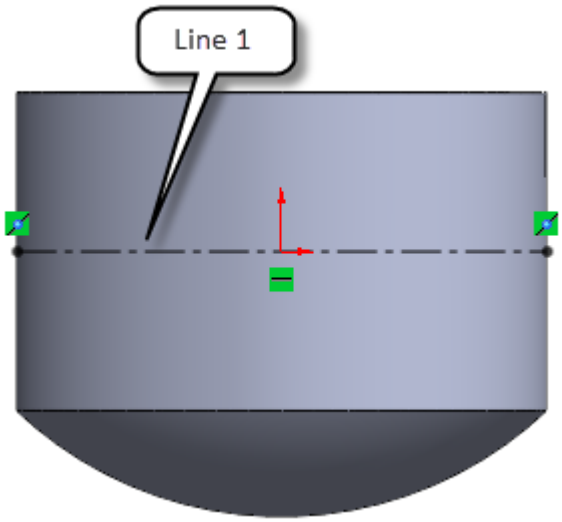


Figure 9.42.: Line 1

Note: *In the **Sketch Text PropertyManager Line 1** appears in the box besides the **Select Edges, Curves, Sketches, Sketch Segments** field.*

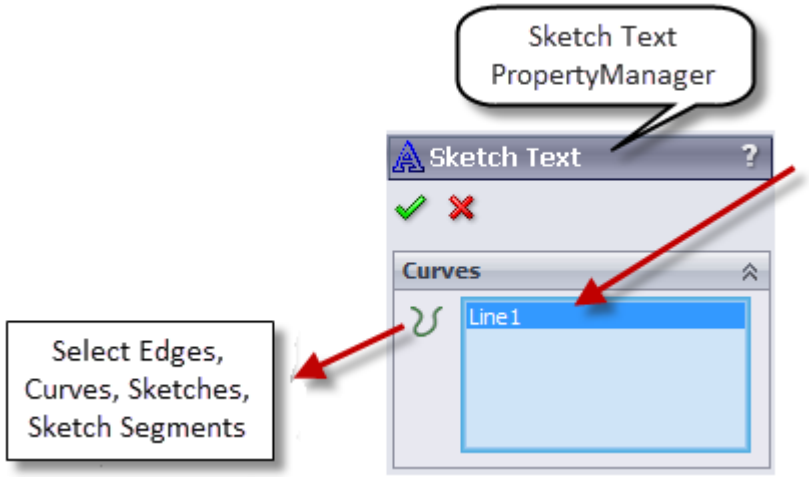


Figure 9.43.: Select Edges, Curves, Sketches, Sketch Segments in the Sketch Text PropertyManager

- **5.1.10.** In the **Sketch Text PropertyManager** under **Text**, type in "CamInstructor" and left click on the mouse **once** on the **Center Align** icon.

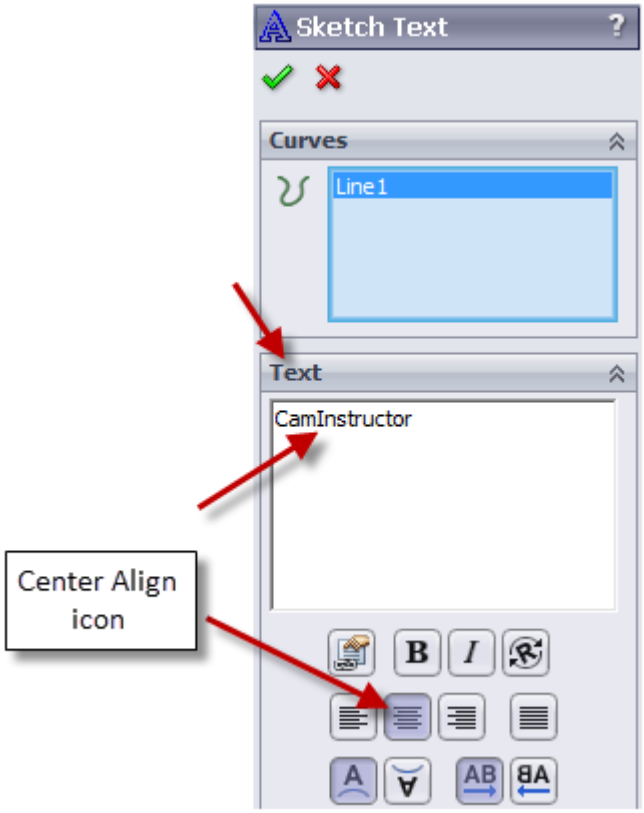


Figure 9.44.: Text and Center Align icon in the Sketch PropertyManager

- **5.1.11**. In the **Sketch Text PropertyManager** click on **OK (green check mark)** to **exit**.

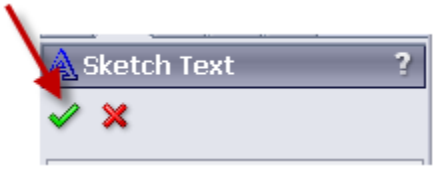


Figure 9.45.: Ok in the Sketch Text PropertyManager

- **5.1.12.** Left click on the mouse **twice** in the **SolidWorks drawing area** to **exit** and **save** the sketch.

- **5.1.13.** Press the **Esc** key on the keyboard to clear your selection.

Note: *In previous projects you exited and save the sketch using the icon on top right corner of the* ***SolidWorks drawing area****. Both ways will provide the* ***same*** *result!!*

In the **SolidWorks drawing area**, the solid will look as shown below.

Figure 9.46.: Solid

5.2. Wrap the Sketch

- **5.2.1.** Left click on the mouse **once** on the **Features tab** to display the **Features CommandManager**.

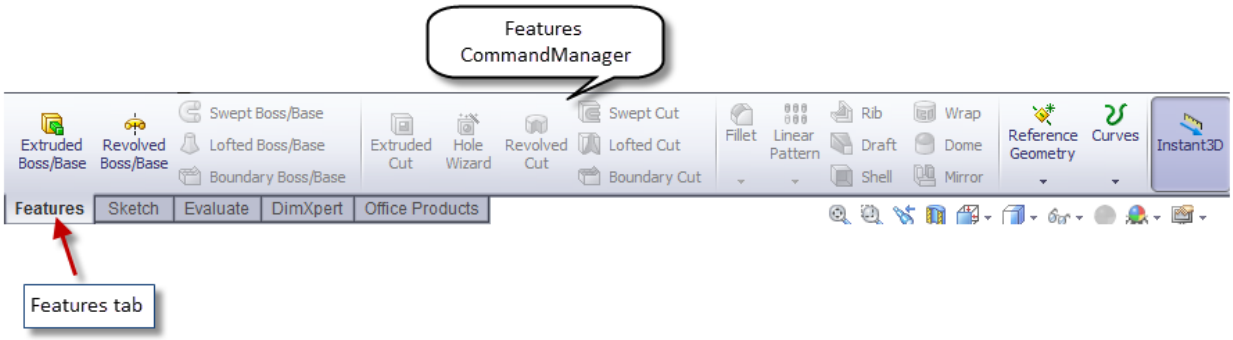


Figure 9.47.: Features tab in the Features CommandManager

- **5.2.2.** Left click on the mouse **once** on the **Wrap** button in the **Features CommandManager**.

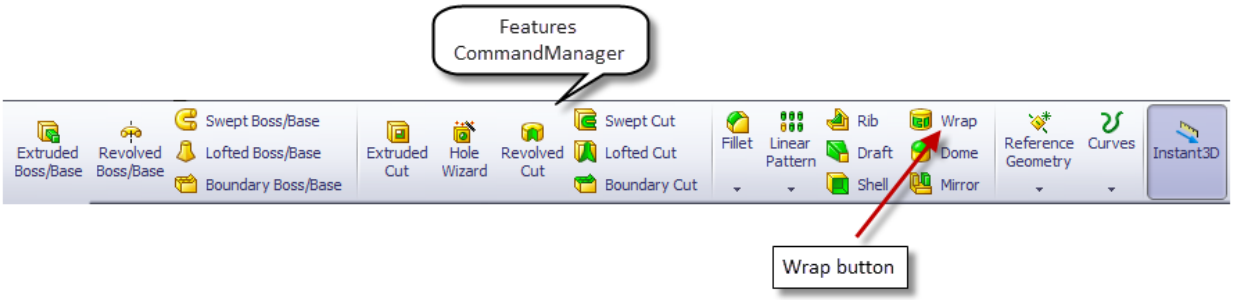


Figure 9.48.: Wrap button in Features CommandManager

*An **Edit Sketch PropertyManager** will appear. The **Edit Sketch PropertyManager** is prompting you to select a plane or an existing sketch to create a features.*

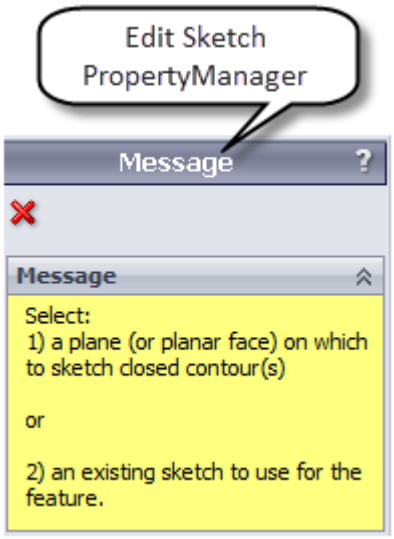


Figure 9.49.: Edit Sketch PropertyManager

- **5.2.3.** Move the cursor to the **SolidWorks drawing area** and left click on the mouse **once** on the "**+**" besides Part1(Default<<Default>_... .

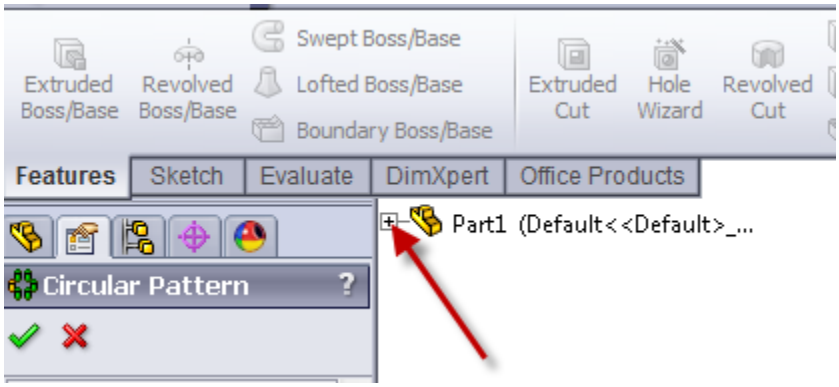


Figure 9.5.: FeatureManager design tree in the SolidWorks drawing area

***Note:** By clicking on the **"+"** you have basically opened up the **FeatureManager design tree** in the **SolidWorks drawing area**.*

- **5.2.4.** In the **FeatureManager design tree** left click on the mouse **once** on **Sketch2**.

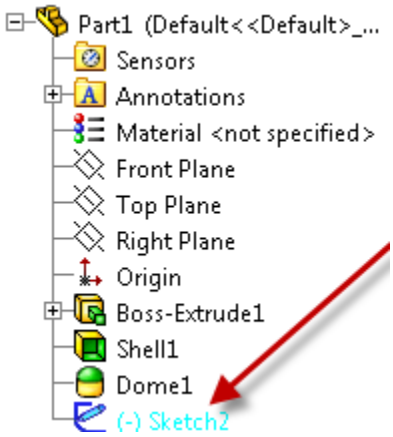


Figure 9.51.: Sketch2 in the FeatureManager design tree

Note: Sketch2 *will highlight in* ***blue****.*

The ***Wrap Feature PropertyManager*** *will appear. The* ***Wrap Feature PropertyManager*** *allows you to sketch onto a face of a solid.*

Note: Sketch2 *will appear under* ***Source Sketch*** *in the* ***Wrap Features PropertyManager****.*

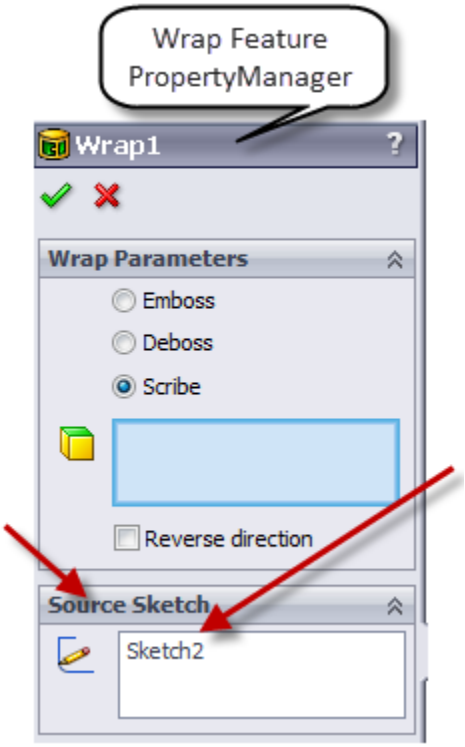


Figure 9.52.: Sketch2 in the Wrap Features PropertyManager

- **5.2.5.** In the **Wrap Features PropertyManager** under **Wrap Parameters** select the radio button besides **Scribe**.

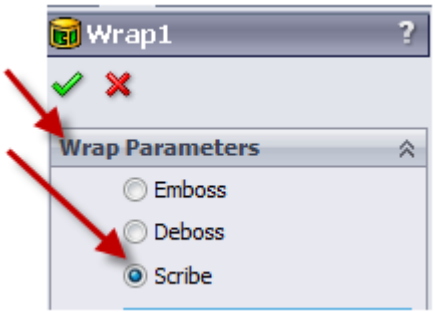


Figure 9.53.: Scribe in the Wrap Features PropertyManager

Note: Scribe *allows you to etch out the sketch on a face*.

- **5.2.6.** Move the cursor to the **SolidWorks drawing area** and left click on the mouse **once** on the **front face** of the extruded solid.

.

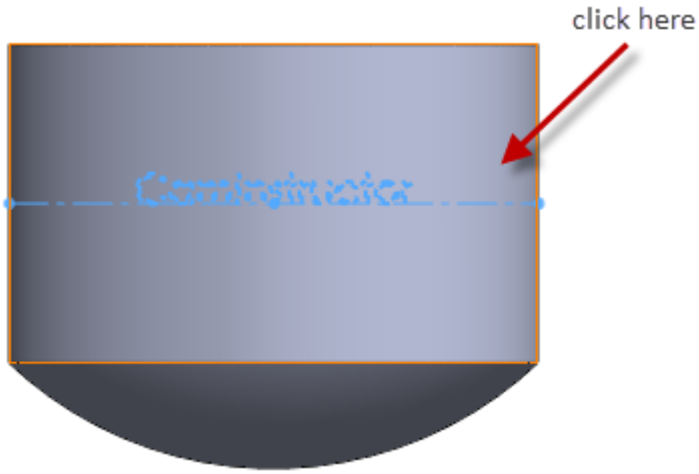


Figure 9.54.: front face of the extruded solid

Note: ***Face <1>*** *will appear in the box besides the* ***Face for Wrap Sketch*** *filed in the* ***Wrap Features PropertyManager****.*

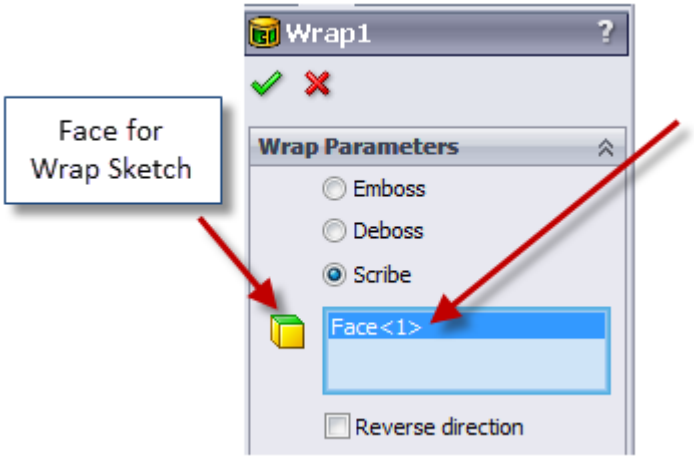


Figure 9.55.: Face for Wrap Sketch field in the wrap Features PropertyManager

- **5.2.7**. In the **Wrap Features PropertyManager** click on **OK (green check mark)** to **exit**.

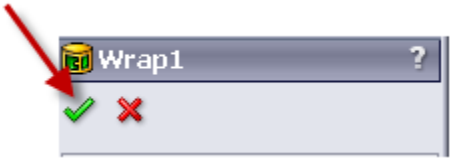


Figure 9.56.: OK in the Wrap Features PropertyManager

- **5.2.8.** Press the **space bar** on the keyboard **once**.

*The **Orientation** window will appear. The **Orientation** window allows you to orient the entities in the **SolidWorks Drawing area** to the view you choose.*

- **5.2.9.** In the **Orientation** window left click on the mouse **twice** on ***Isometric**.

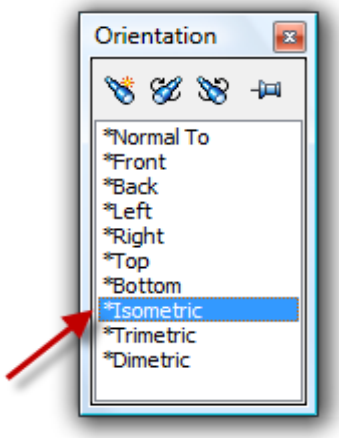


Figure 9.57.: *Isometric in the Orientation window

In the **SolidWorks drawing area**, the completed solid will look as shown below.

Figure 9.58.: Solid

6. Save the Part

6.1. Left click on the mouse **once** on the **Save** icon in the **Menu** bar.

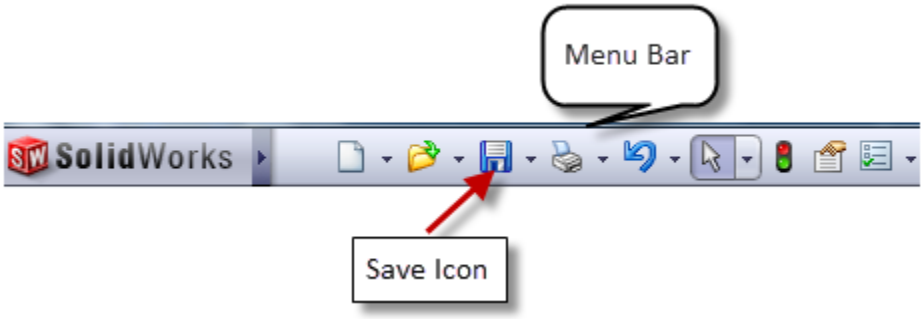


Figure 9.59.: Save icon in the Menu Bar

6.2. Choose an appropriate location to **save** the file and type **Project M9** besides **File name** and, make sure **Save as type** is set to **Part (*.prt, *.sldprt).**

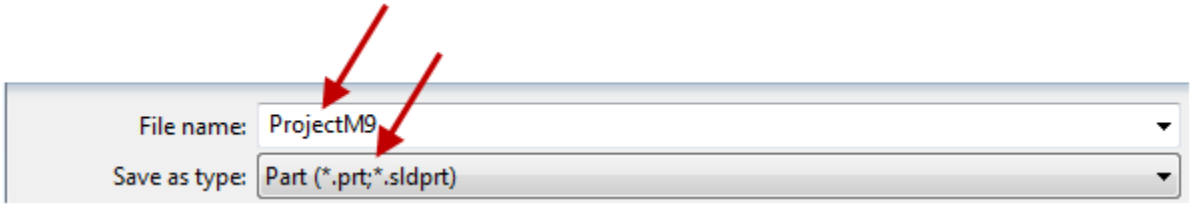


Figure 9.6.: File Name and Save as type

6.3. Left click on the mouse **once** on **Save**.

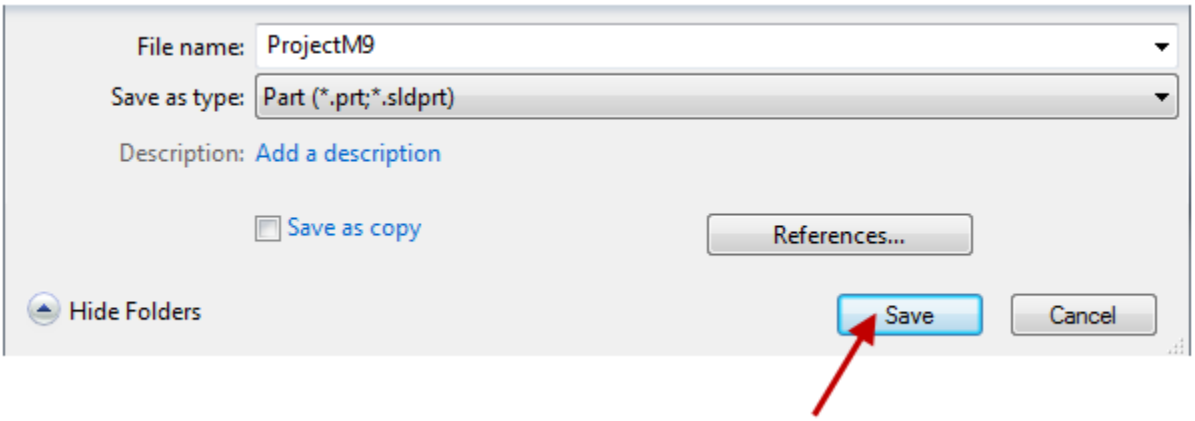
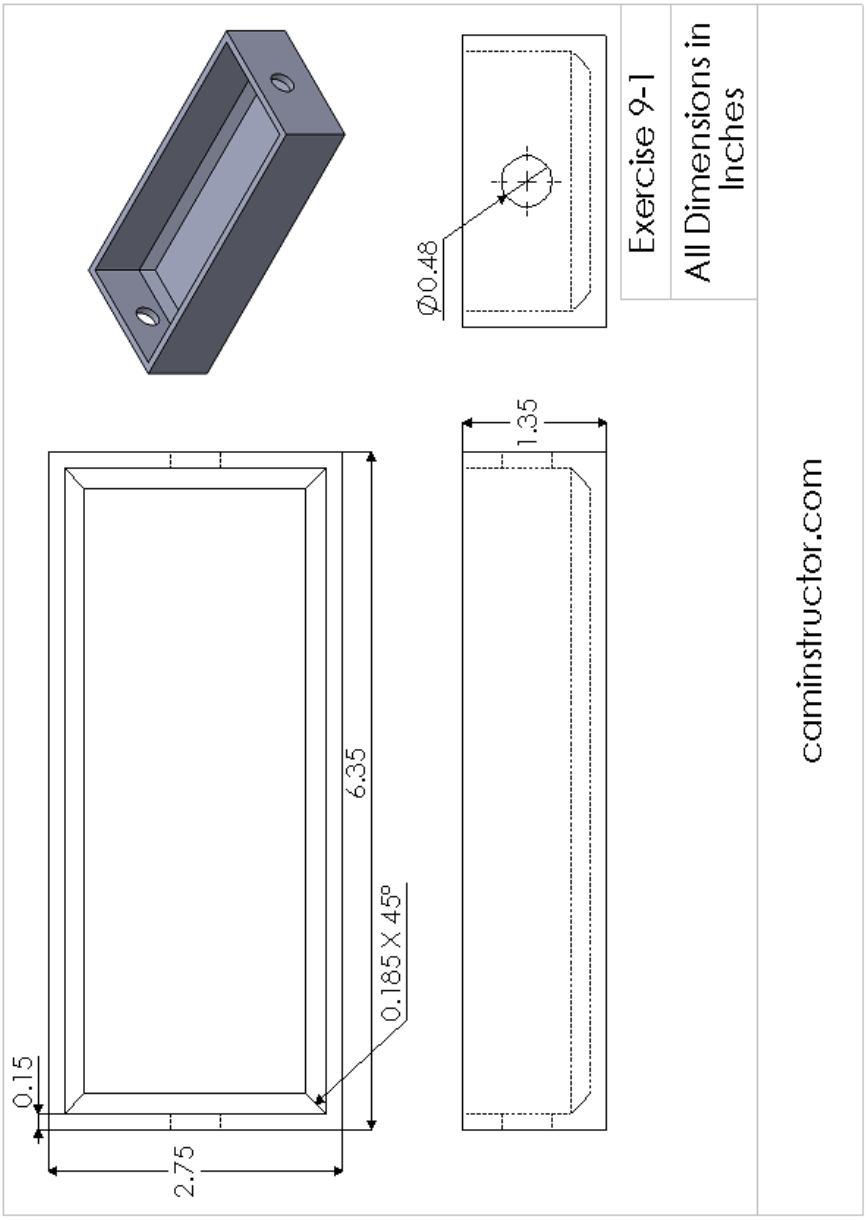
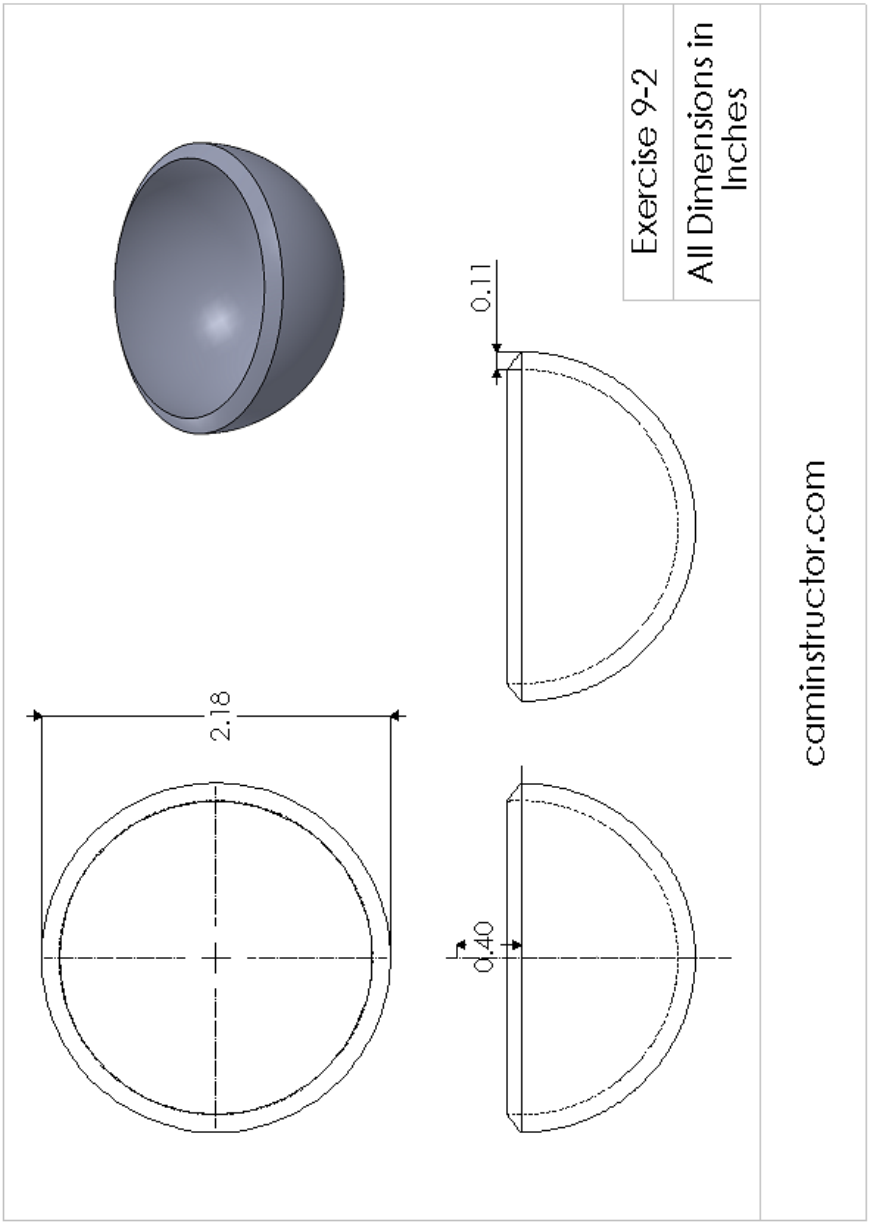


Figure 9.61.: Save

Congrats! You have just completed **Project M9** and your ninth part modeling in SolidWorks.


Exercise 9-1
All Dimensions in Inches
caminstructor.com
Ø0.48
1.35
6.35
0.185 X 45°
0.15
2.75


Exercise 9-2
All Dimensions in Inches
caminstructor.com
0.11
2.18
0.40