SOLIDWORKS® 2016 Intermediate Skills

Expanding on Solids, Surfaces, Multibodies, Configurations, Drawings, Sheet Metal and Assemblies

Paul Tran CSWE, CSWI

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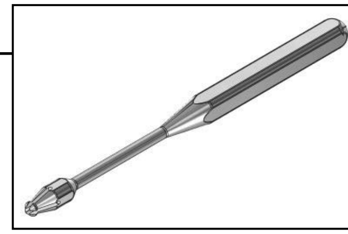
CHAPTER 2

Sketching

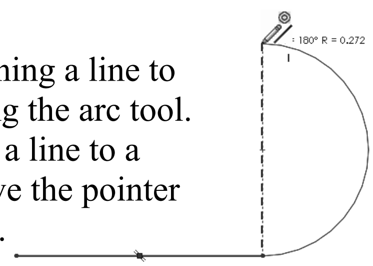


Sketching - Overview

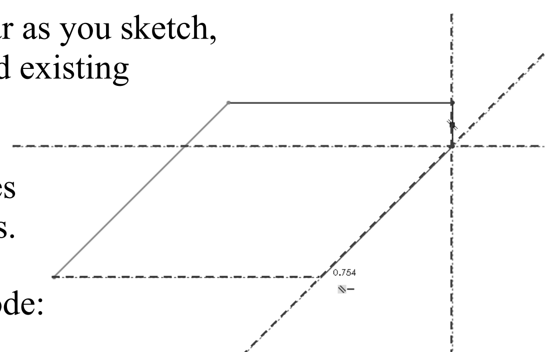
Handle

Most features in SOLIDWORKS start with a sketch. The sketch is the basis for a 3D model. You can create a sketch on any of the default planes (Front Plane, Top Plane, and Right Plane) or a created plane.

There are two modes for sketching in 2D: click-drag or click-click. The click-drag method will create a single entity each time, but the click-click creates multiple, connecting lines.



While sketching the lines you can transition from sketching a line to sketching a tangent arc, and vice versa, without selecting the arc tool. Simply press the A key on the keyboard to switch from a line to a tangent arc – OR - start the line with the first click, move the pointer outward and back to the starting point, then away again.

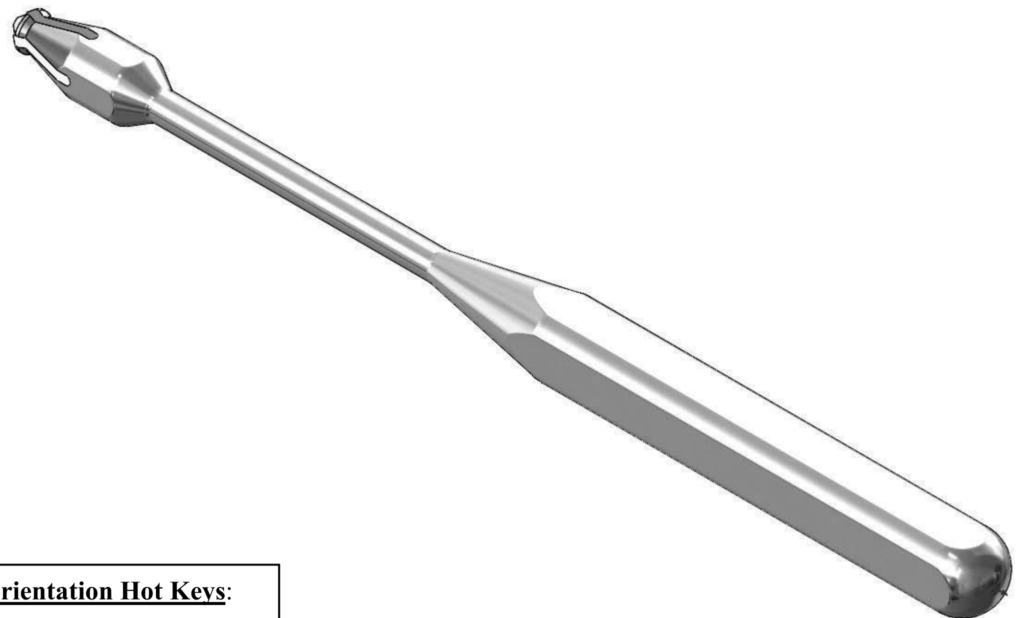


Inferencing lines are dotted lines that appear as you sketch, displaying relations between the pointer and existing sketch entities or model geometry. When your pointer approaches highlighted cues such as midpoints, the inferencing lines guide you relative to existing sketch entities.

There are 2 types of Snaps in the sketch mode:
Sketch Snap and Quick Snap.
Each Sketch Snap allows you to automatically snap to selected entities as you sketch. By default, all Sketch Snaps except Grid are enabled.

Quick Snaps are instantaneous, single operation Sketch Snaps. Sketching any sketch entity (such as a line) from start to finish is a single operation.

Sketching
Handle



View Orientation Hot Keys:

Ctrl + 1 = Front View
Ctrl + 2 = Back View
Ctrl + 3 = Left View
Ctrl + 4 = Right View
Ctrl + 5 = Top View
Ctrl + 6 = Bottom View
Ctrl + 7 = Isometric View
Ctrl + 8 = Normal To Selection

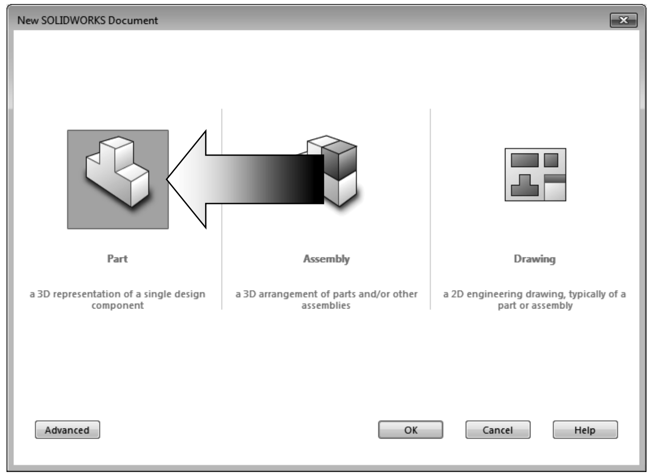
Dimensioning Standards: **ANSI**

Units: **INCHES** – 3 Decimals

Tools Needed:

Insert Sketch	Line	Circle
Straight Slot	Centerline	Smart Dimension
Extruded Boss	Extruded Cut	Chamfer

1. Starting a new part document:

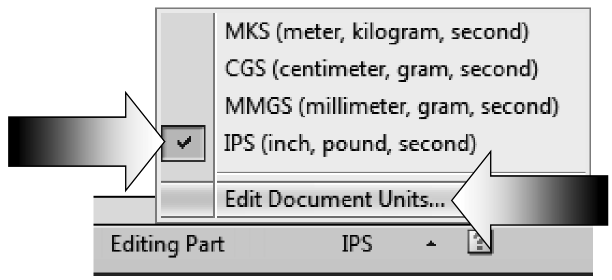


- Select **File / New**.

- The “Novice” dialog box by default displays three template options: **Part, Assembly**, and **Drawing**.

- Select the **Part** template and click **OK** (arrow).

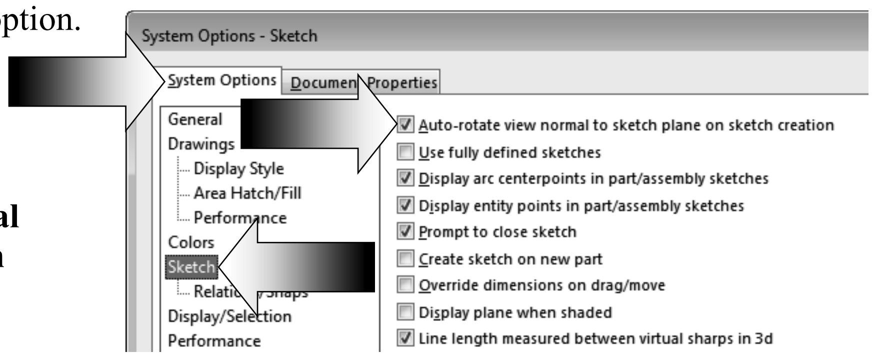
2. Changing the System Options:



- Click the small arrow at the bottom right corner of the screen and select **IPS** for **Inch, Pound, Second**.

- Select **Tools, Options**.

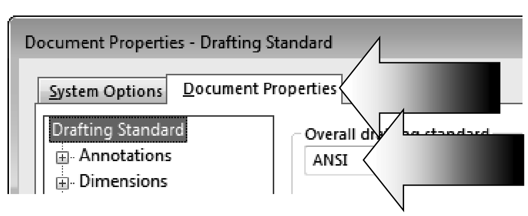
- Click the **Sketch** option.



- Enable the check-box for: **Auto-rotate view normal to sketch plane on sketch creation**.

- Switch to the **Document Properties** tab.

- Click the **Drafting Standard** option and select the **ANSI** standard from the list.



- Click **OK**.

3. Creating the Parent sketch:

Sketching in SOLIDWORKS is the basis for creating features. Features are the basis for creating parts, which can be put together into assemblies. Sketch entities can also be added to drawings.

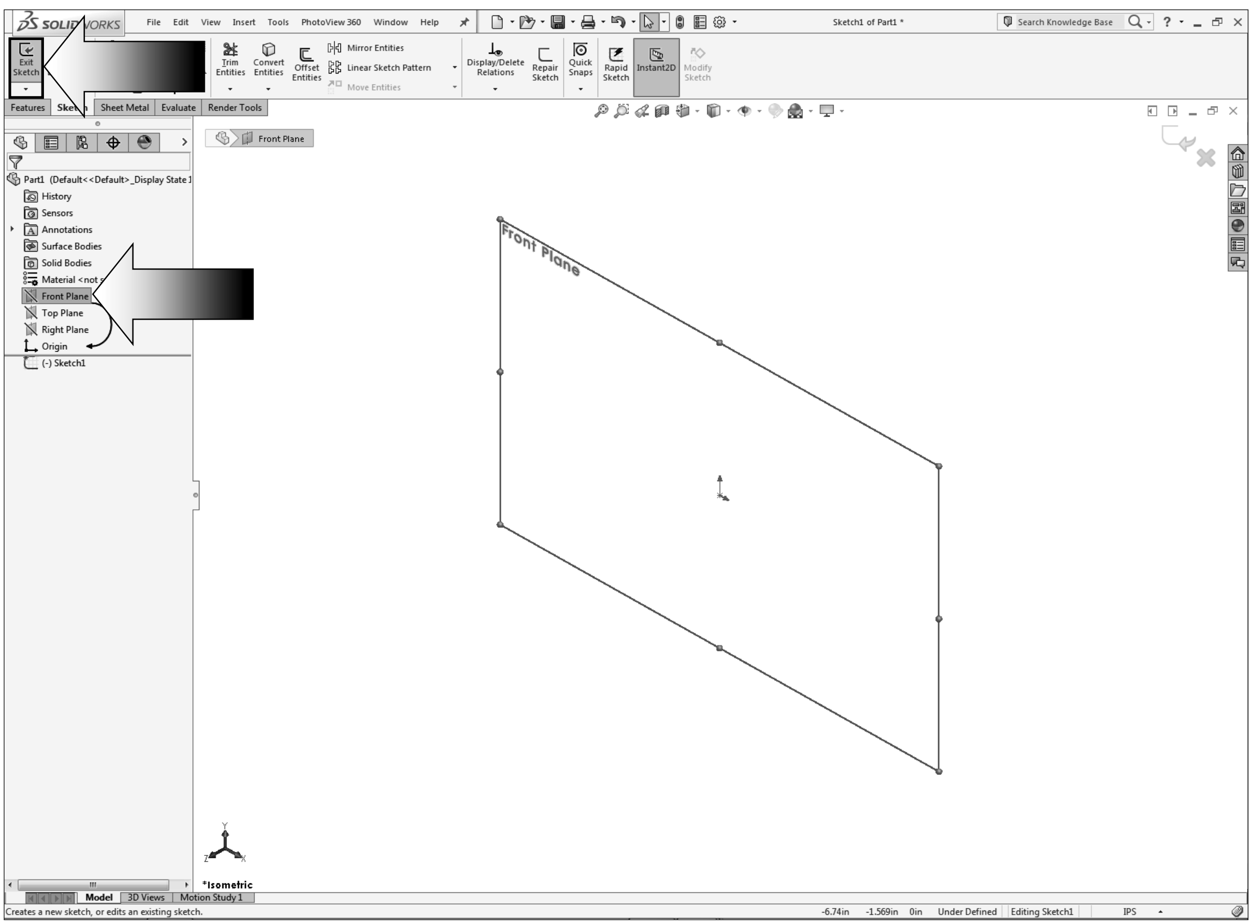
SOLIDWORKS sketch entities can snap to points (endpoint, midpoints, intersections, etc.) of other sketch entities. With Quick Snaps, you can filter the types of sketch snaps that are available.

Inferencing displays relations by means of dotted inferencing lines, pointer display, and highlighted cues such as endpoints and midpoints.

The sketch status appears in the window status bar. Colors indicate the state of individual sketch entities.

- The 1st sketch in a part document is considered the parent sketch.

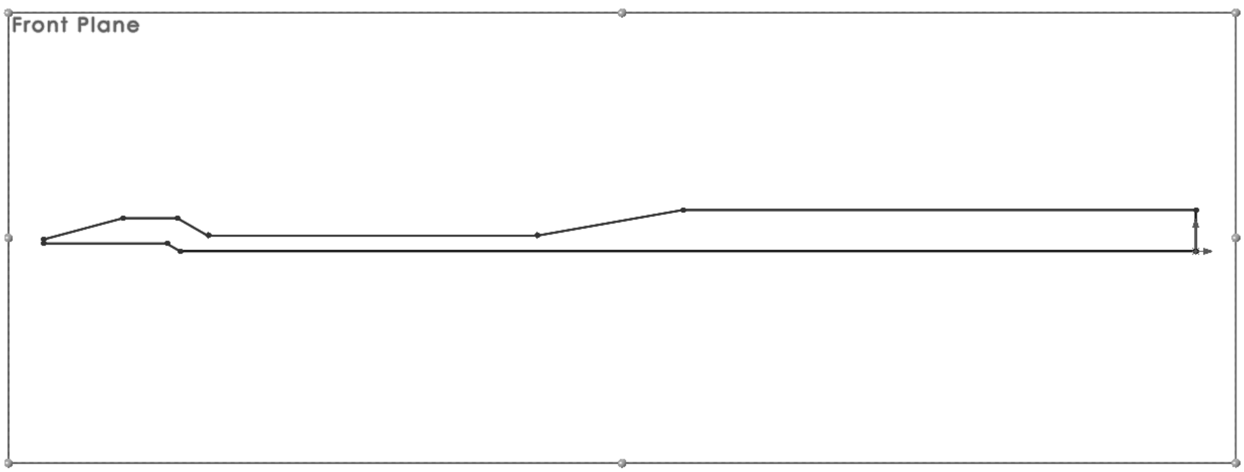
- Select the **Front** plane and open a **new sketch** (click the Sketch Pencil).



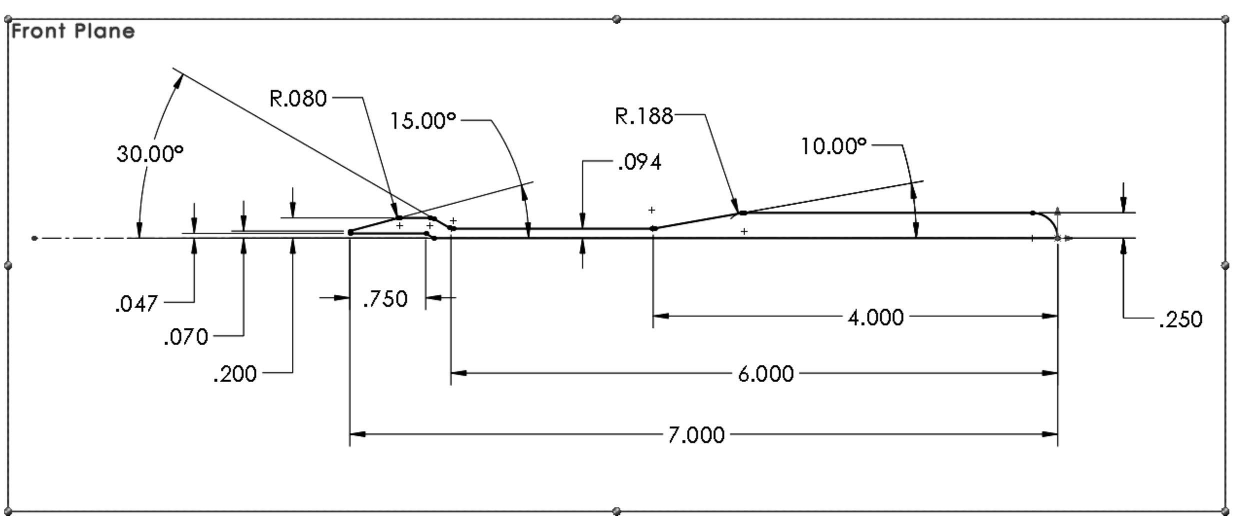
- Select the **Line** command from the Sketch toolbar.

- Use the **Click-Click** method and start the first line at the Origin, move upward to make a vertical line, and then a horizontal line as indicated.

- Continue with sketching the rest of the lines as shown below.

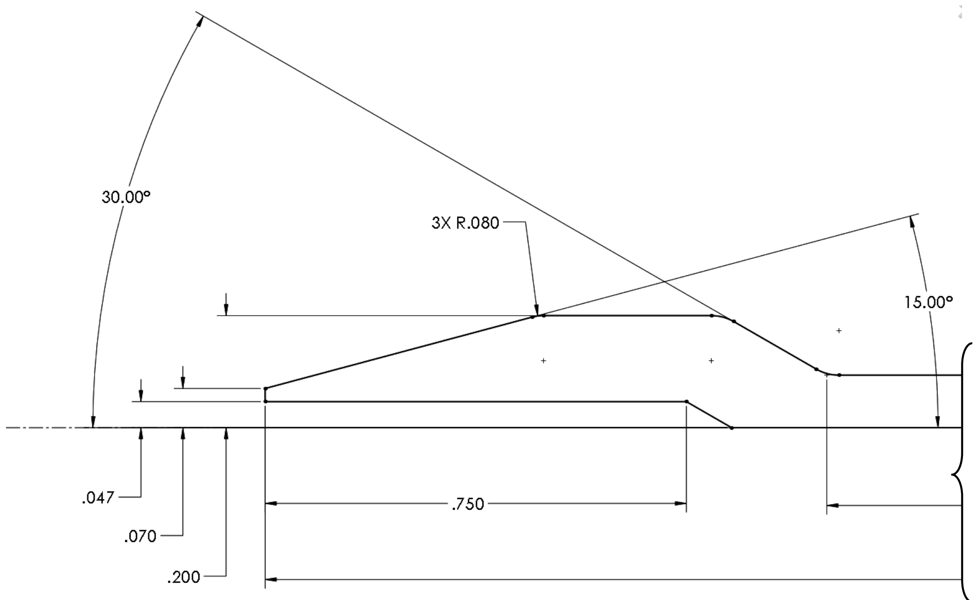


- Select the **Smart Dimension** command , add the dimensions shown More...



- It is better to add the Sketch Fillets after the sketch is fully defined.

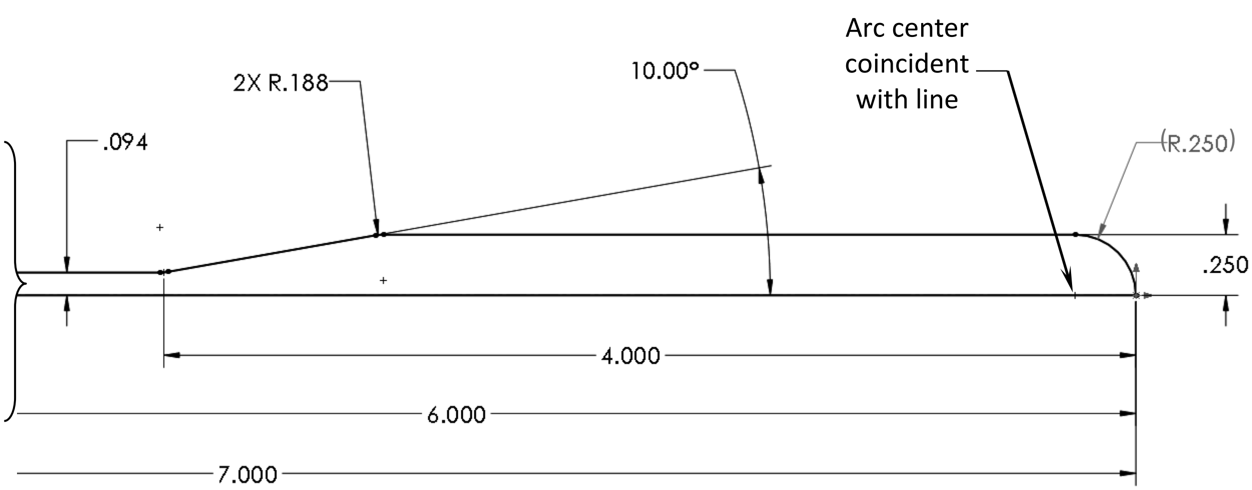
- Add the dimensions to the left end of the sketch as indicated below.



Left end of the sketch

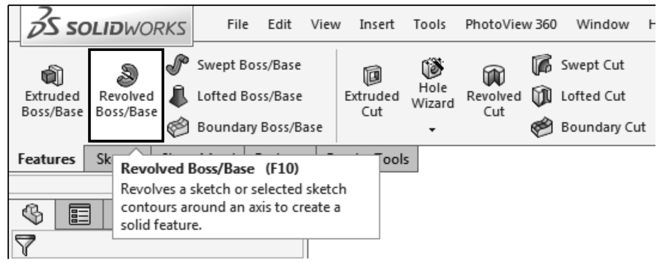
- Add the dimensions to the right end of the sketch as shown.

- The dimension R.250 is a reference dimension and shown in gray color.



Right end of the sketch

4. Revolving the parent sketch:

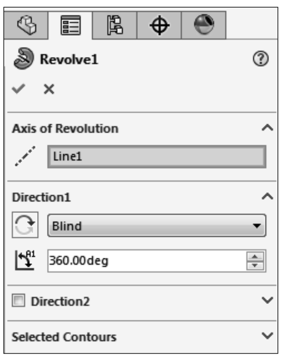


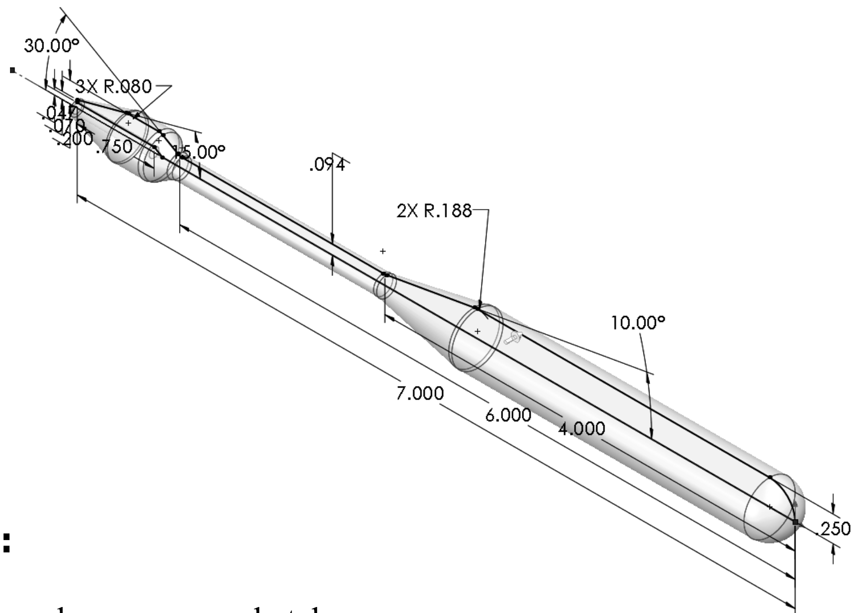
- Switch to the **Features** tool tab.

- Select the revolved **Boss/Base** command.

- Use the default **Blind** type and **360º** angle.

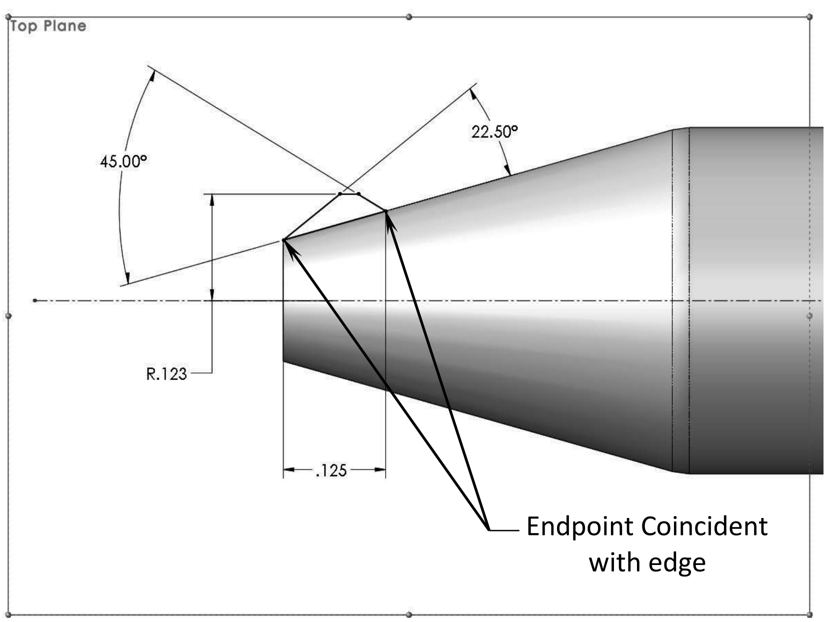
- Click **OK**.





5. Adding the tip detail:

- Select the **<u>Top</u>** plane and open a new sketch.



- Sketch the profile shown on the right.

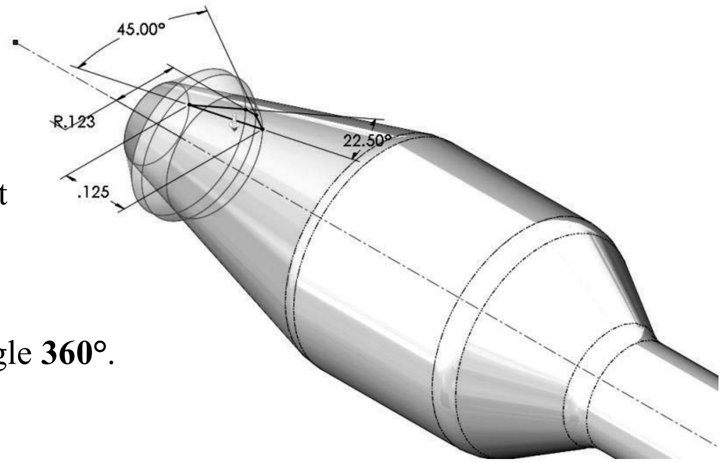
- Sketch a horizontal centerline to use as the revolve line.

- Add the relations and the dimensions as indicated.

- Be sure to fully define the sketch before revolving it.

6. Revolving the sketch:

- Select **Revolved Boss/Base**.

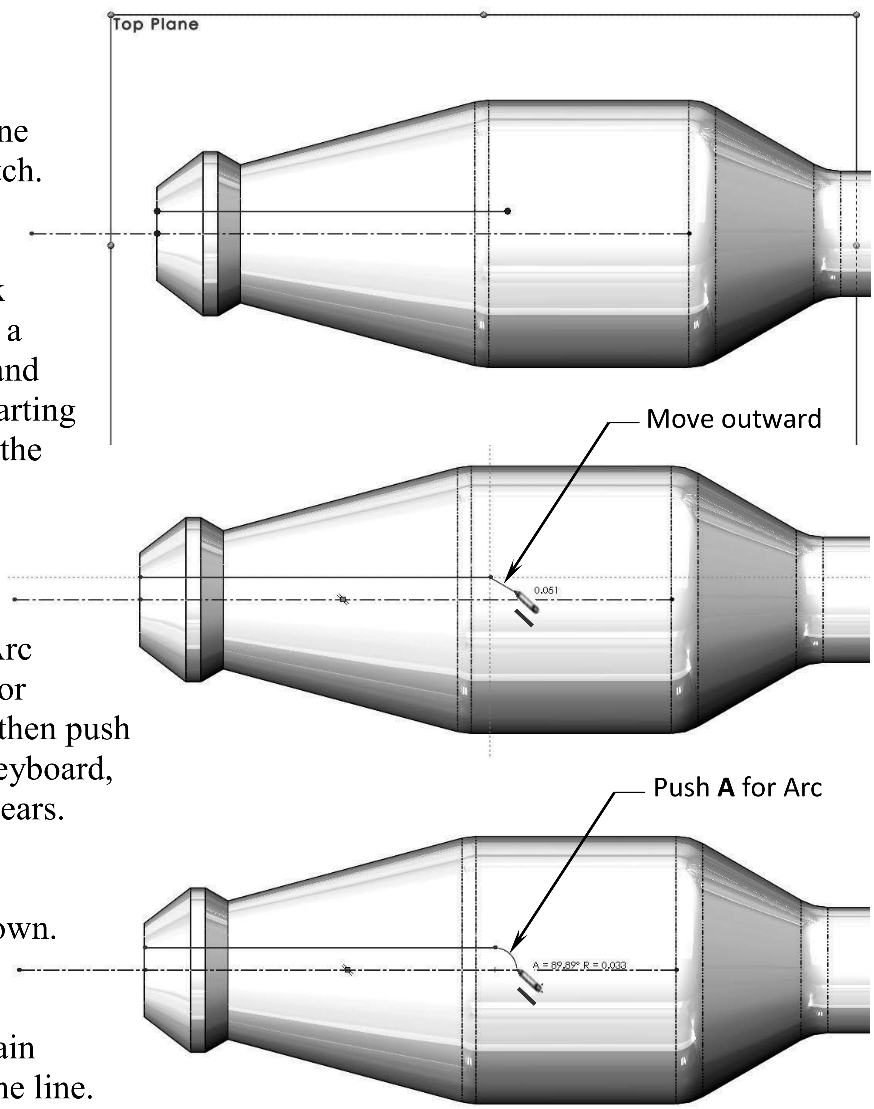


- Use the default **Blind** type.

- Revolved Angle **360°**.

- Click **OK**.

7. Transitioning from Line-to-Arc:



- Select the <u>**Top**</u> plane and pen a new sketch.

- Use the **click-click** method and sketch a short vertical line and a horizontal line starting at the midpoint on the left side.

- To change from a line to a Tangent Arc first move the cursor outward as shown then push the <u>**A**</u> key on the keyboard, the tangent arc appears.

- Make an arc as shown.

- Push the A key again to switch back to the line.

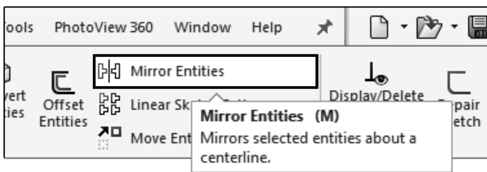
8. Mirroring in sketch mode:

Mirror Entities

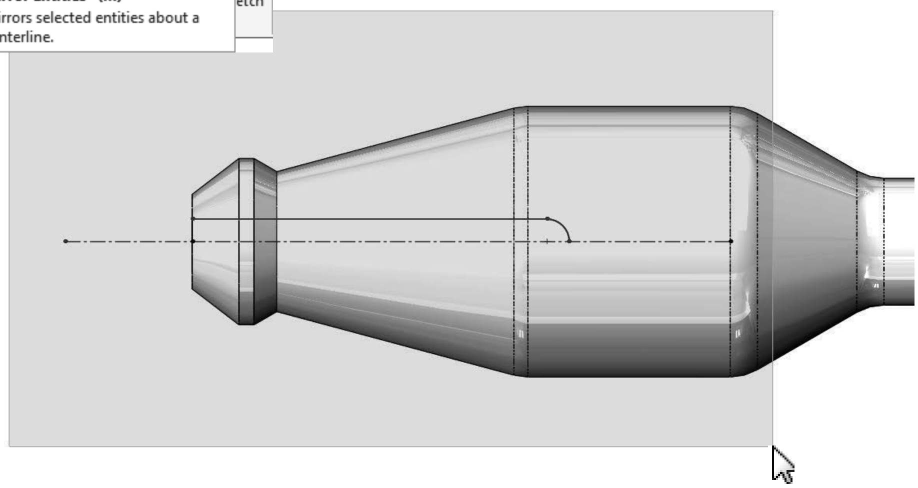
Mirrors selected entities about a centerline, a line, or a model edge.
A Symmetric relation is added to each mirrored entity.

- Push **Esc** to exit the line command.

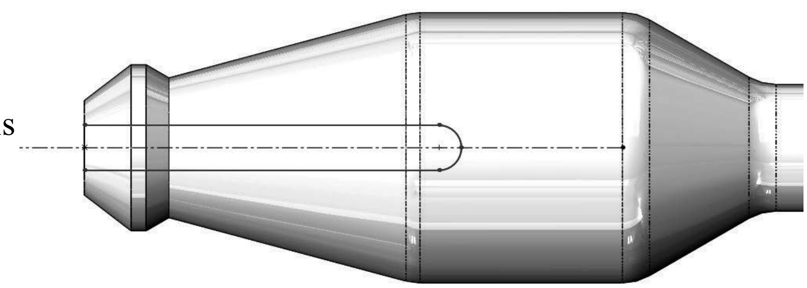
- Box-Select all sketch entities and click the **Mirror Entities** command.

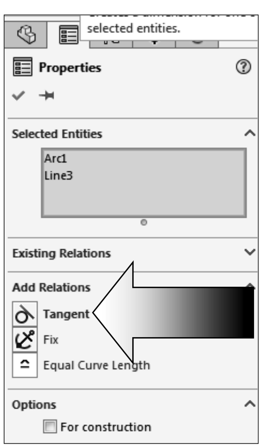


- All selected entities are mirrored about the horizontal centerline.

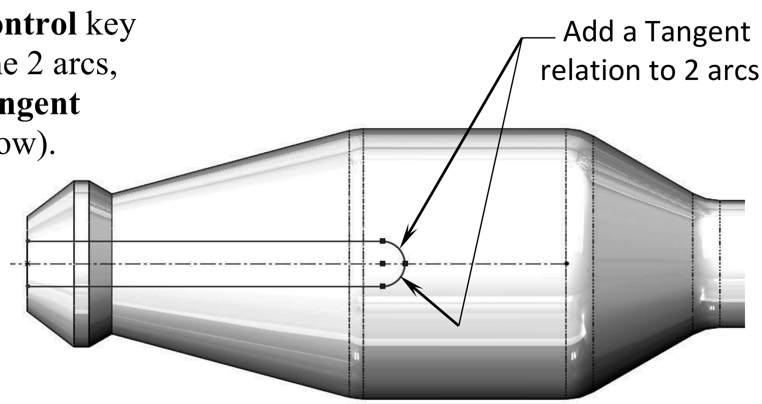


- A **Symmetric** relation is added to each mirrored entity.



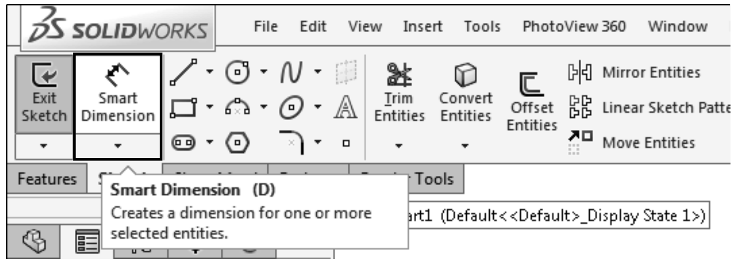


- Hold the **Control** key and select the 2 arcs, click the **Tangent** relation (arrow).



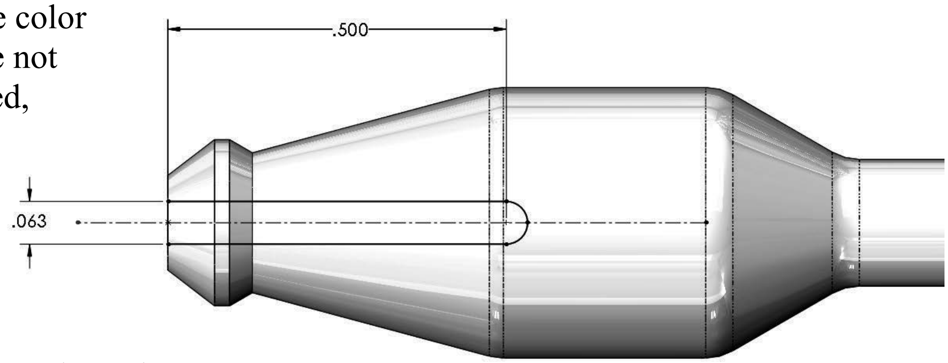
- Click **OK**.

9. Adding dimensions:



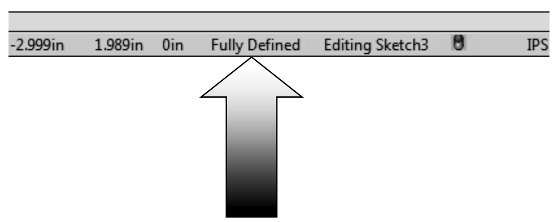
- Use Smart Dimension tool to specify the size and location for each sketch entity.

- The sketch entities appear in blue color when they are not yet constrained, but turn black when relations or dimensions are added.



- Add the 2 dimensions shown.

- The Status of the sketch is displayed at the lower right corner of the screen.

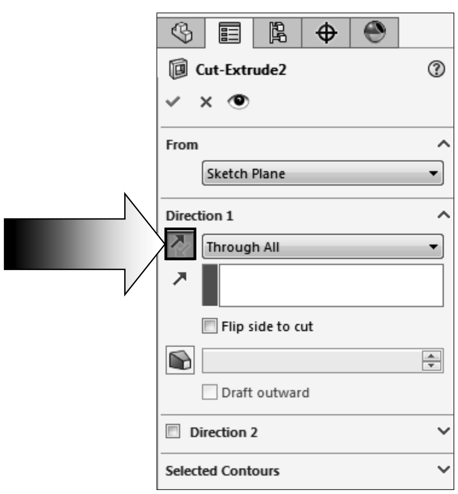


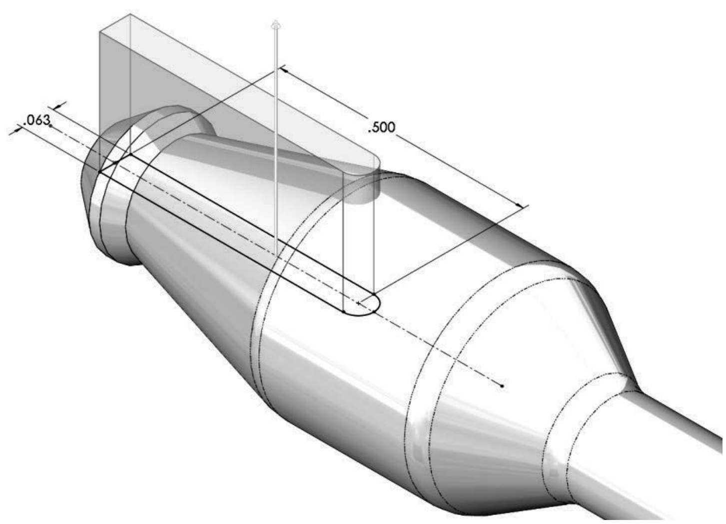
10. Extruding a cut:

- Switch to the Features tool tab and click **Extruded Cut**.

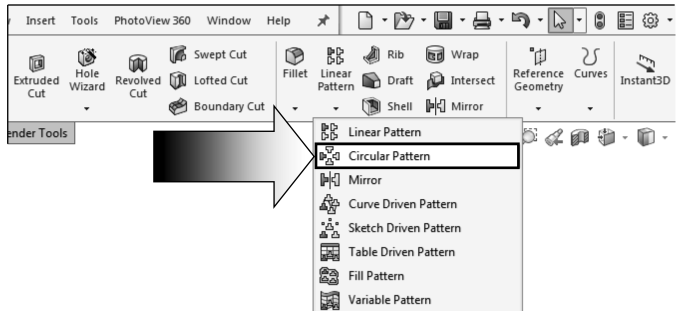
- Select the **Through All** condition from the list and click **Reverse** direction.

- Click **OK**.



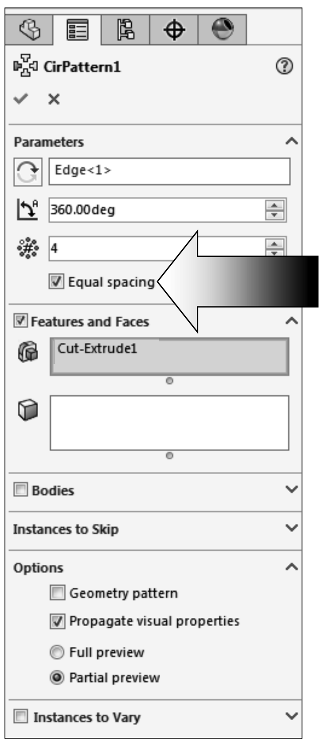


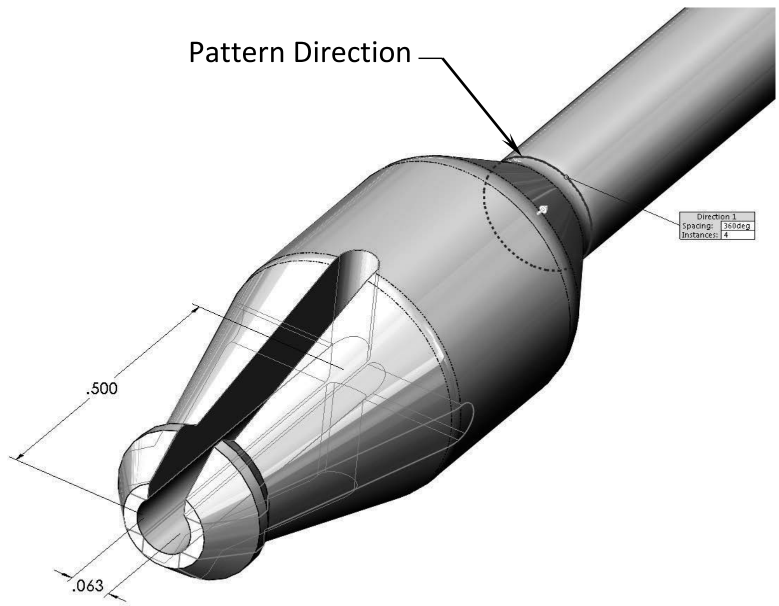
11. Creating a Circular Pattern:



- Circular patterns create multiple instances of one or more features and space them uniformly around an axis.

- Select the **Circular Pattern** command below the Linear Pattern drop down.





- For Pattern Direction, select one of the **circular edges** in the model.

- Enable the **Equal Spacing** checkbox (arrow).

- Enter **4** for Number of Instances.

- Select the **Cut-Extrude1** either from the Feature tree or from the graphics area. The preview graphics of the 4 instances appears.

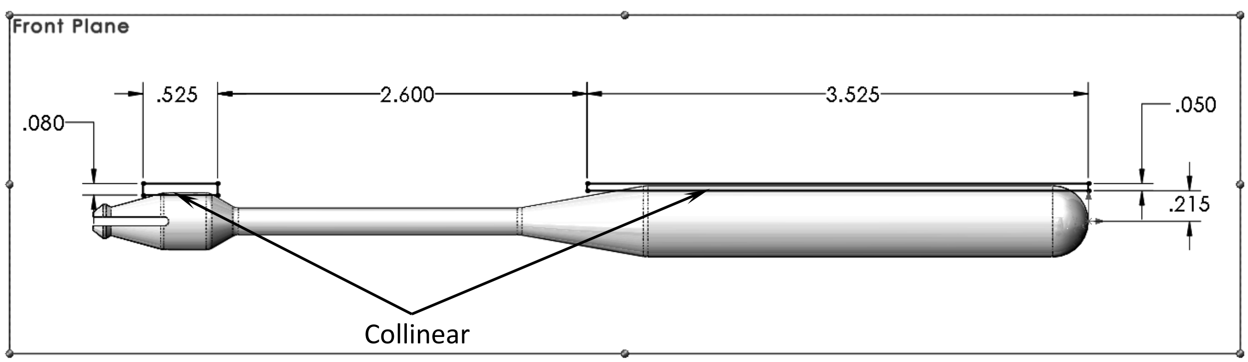
- Click **OK**.

12. Adding other cut features:

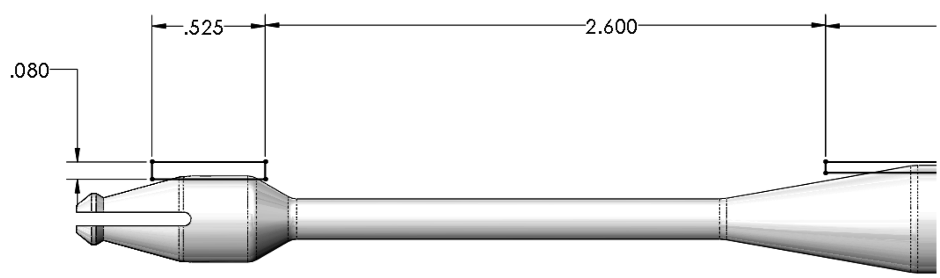
- The flat features not only provide better grips but also help keep the device from rolling around.

- Select the **Top** plane and open a new sketch.

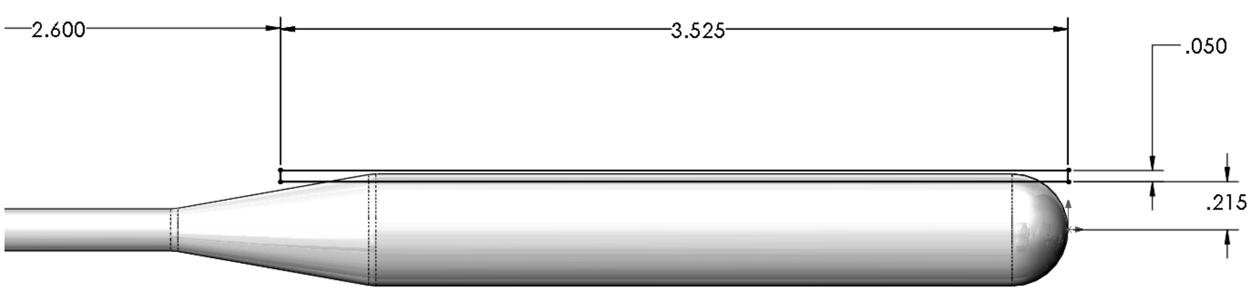
- Sketch **2 Corner Rectangles** at both ends of the model.



- Add the dimensions and relations shown to fully define the sketch.



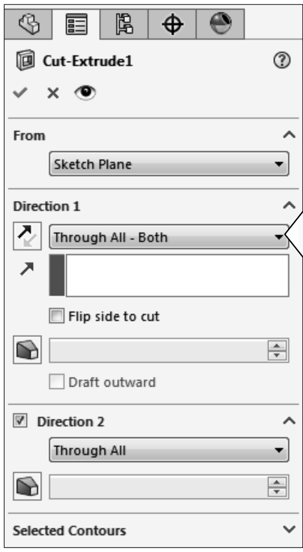
Left end of the sketch

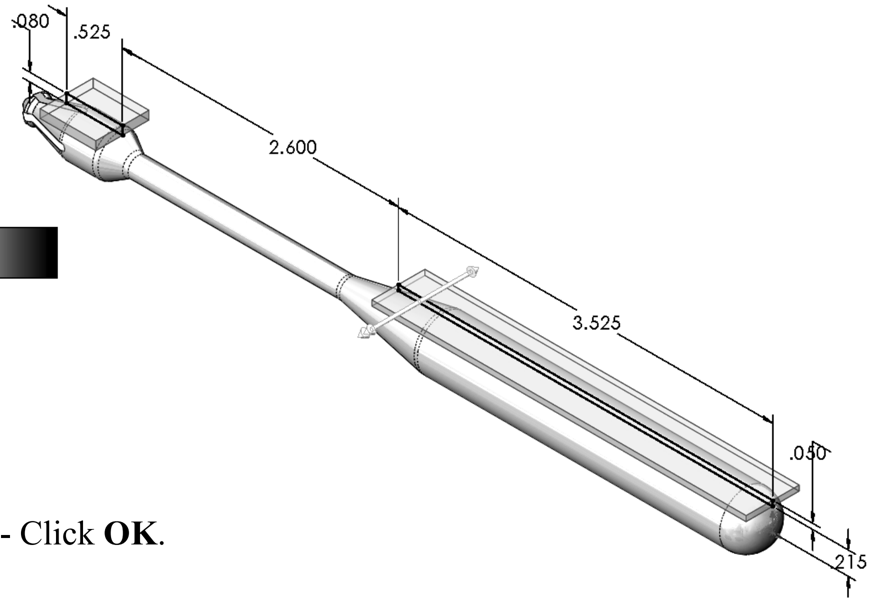


Right end of the sketch

- Switch to the **Features** tool tab and click the **Extruded Cut** command.

- Select the **Through All – Both** from the list to cut through both directions.





- Click **OK**.

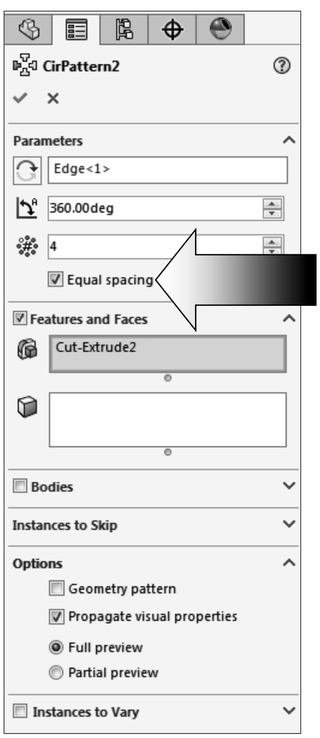
13. Creating another Circular Pattern:

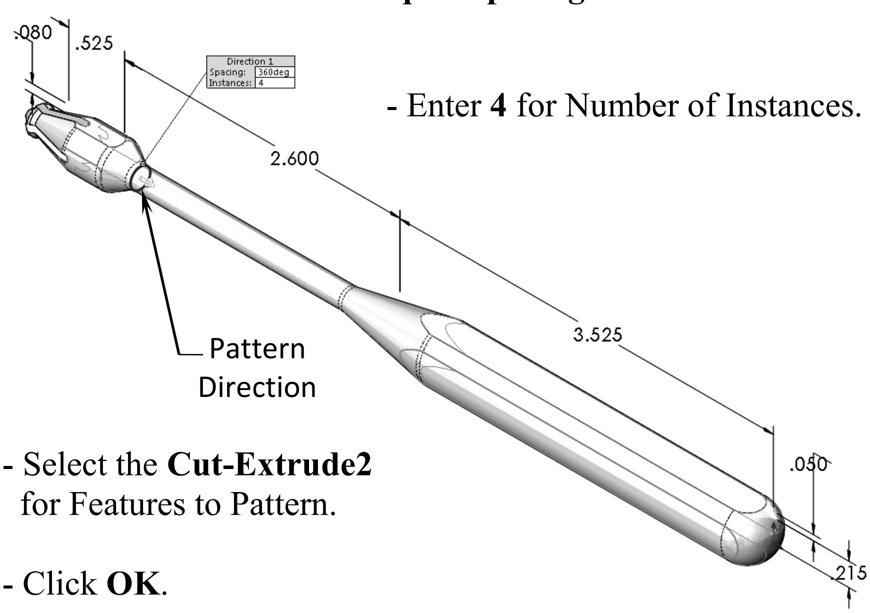
- Select the Circular Pattern command below the Linear Pattern option.

- For Pattern Direction, select one of the **circular edges** of the model.

- Enable the **Equal Spacing** checkbox.

- Enter **4** for Number of Instances.



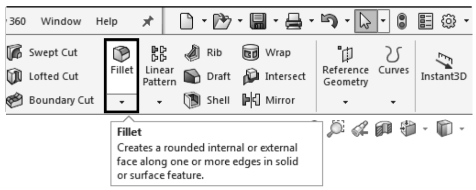


- Select the **Cut-Extrude2** for Features to Pattern.

- Click **OK**.

14. Adding a .032" Constant Size Fillet:

- Select the **Fillet** command  from the Features tool tab.

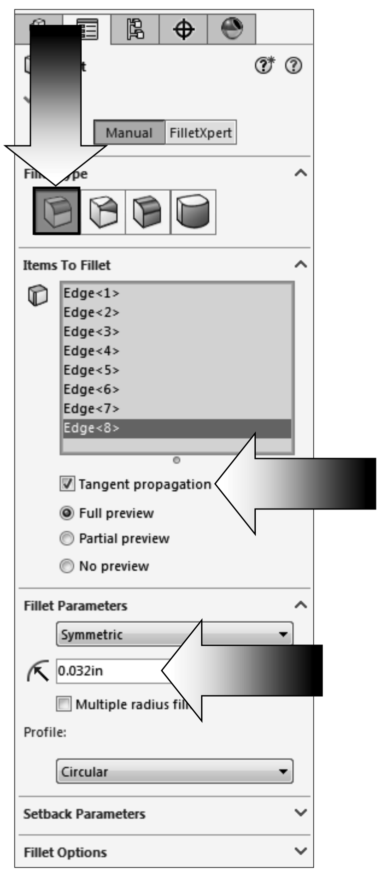


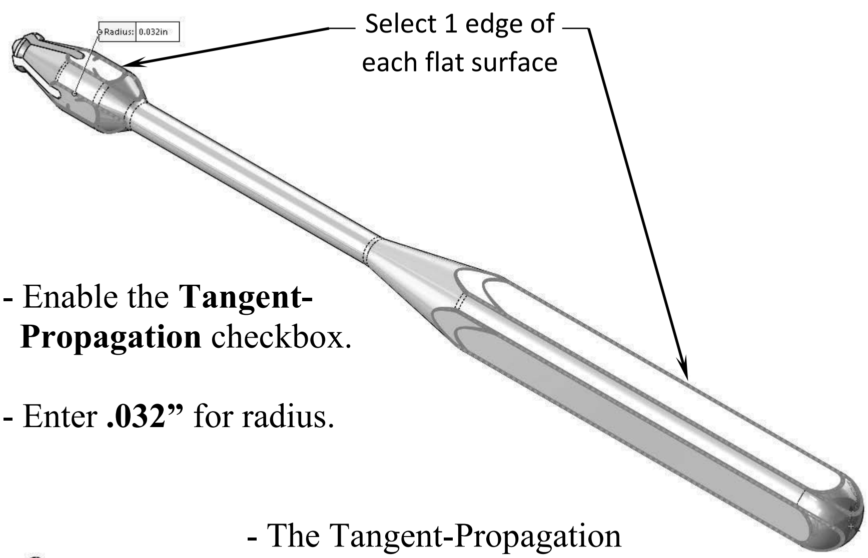
- The **Constant Size Radius** is the default type.

- Select 1 edge for each flat feature, total of **8 edges**.

Constant Size Fillet

Creates fillets that have a constant size for the entire length of the fillet.

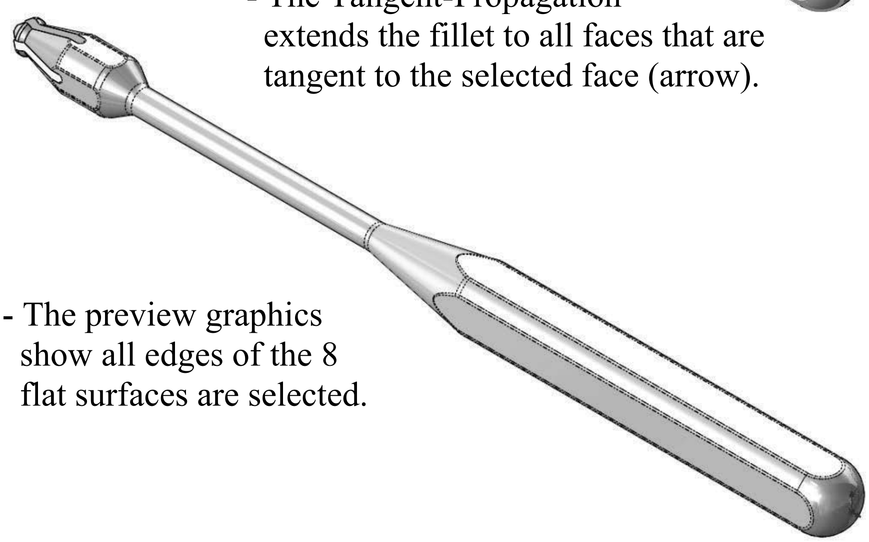




- Enable the **Tangent-Propagation** checkbox.

- Enter **.032"** for radius.

- The Tangent-Propagation extends the fillet to all faces that are tangent to the selected face (arrow).



- The preview graphics show all edges of the 8 flat surfaces are selected.

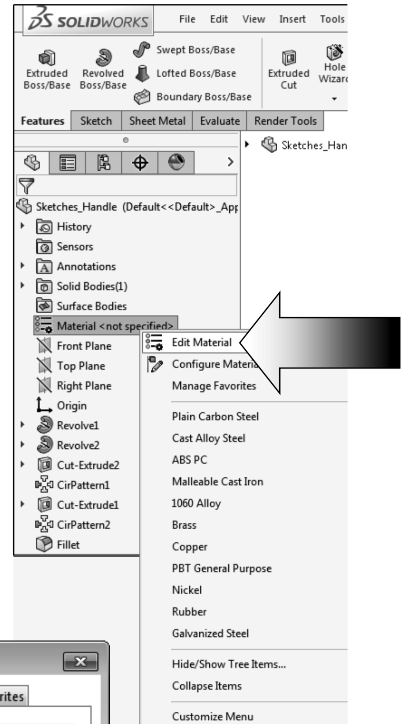
- Click **OK**.

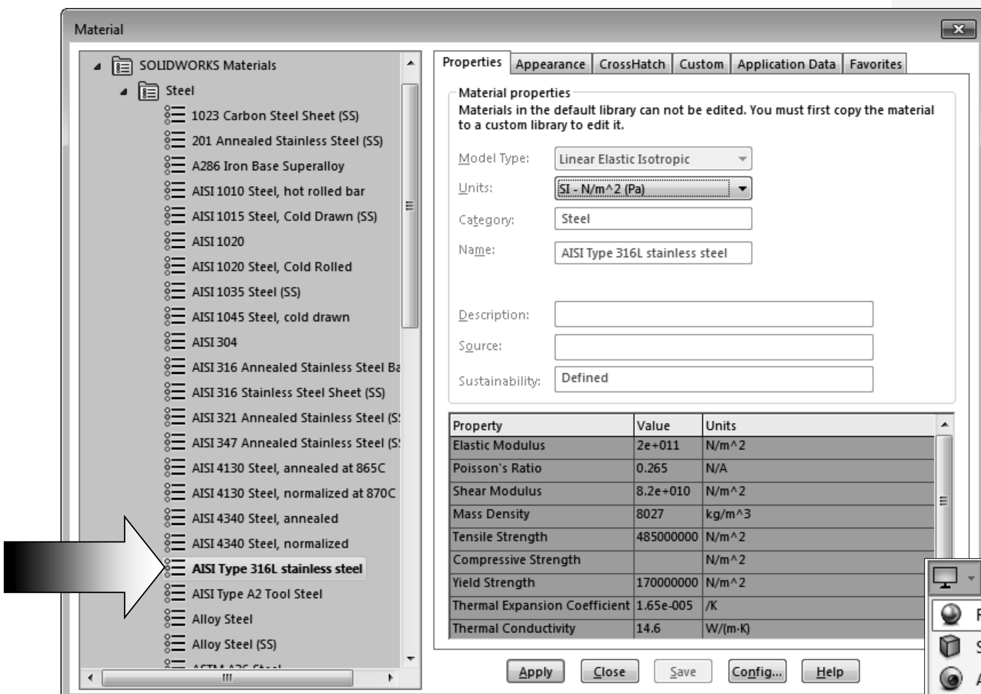
15. Assigning material to the model:

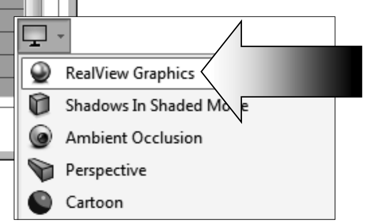
- SOLIDWORKS has two sets of properties, visual and physical (mechanical). The response of a part when loads are applied to it depends on the material assigned.

- Right click **Material** on the FeatureManager tree and select **Edit Material**.

- In the SOLIDWORKS Materials folder, expand the **Steel** sub folder.







- Select the material **AISI Type 316L Stainless Steel** from the list (arrow).

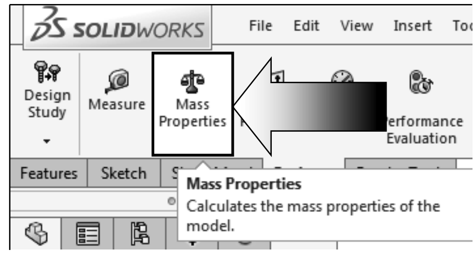
- Click **Apply** and **Close**.

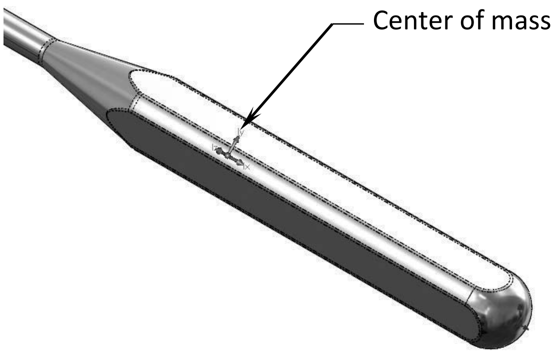
- Enable the RealView Graphics option to enhance the model display if applicable.

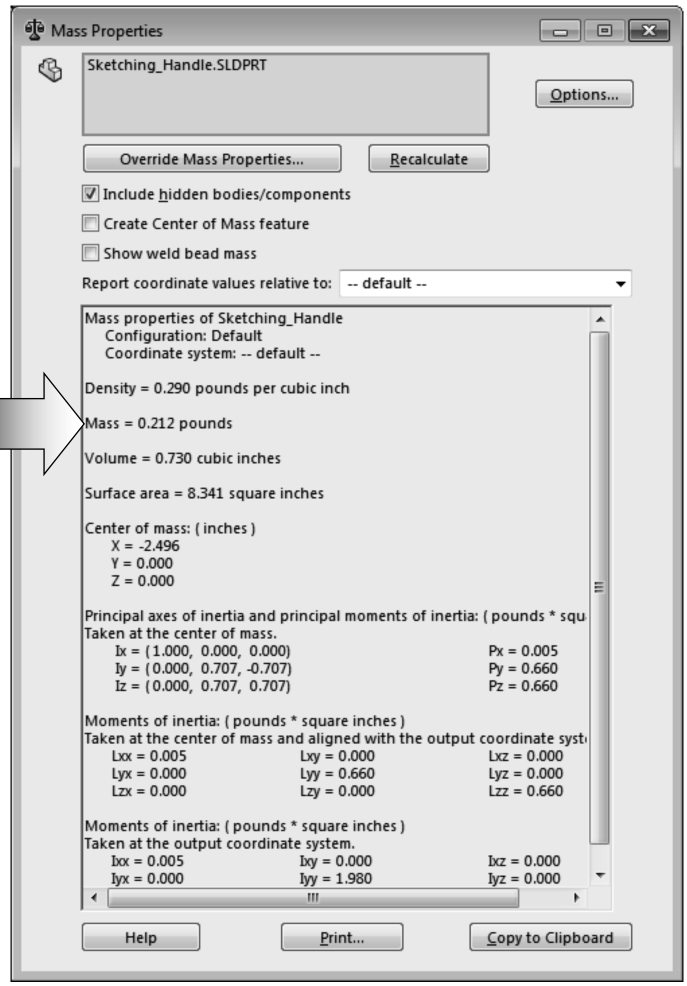
16. Calculating the Mass of the model:

- Switch to the **Evaluate** tab.

- Click the **Mass Properties** command (arrow).







- In the graphics area, a single-colored triad indicates the principal axes and center of mass of the model.

- The mass of the model based on the selected material is: **.212 pounds**.

17. Saving your work:

- Select File / Save As.

- Enter **Sketcing_Hanlde.sldprt** for the name of the part.

- Click **Save**.