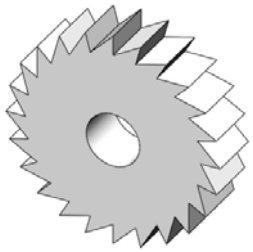
Section 1.2

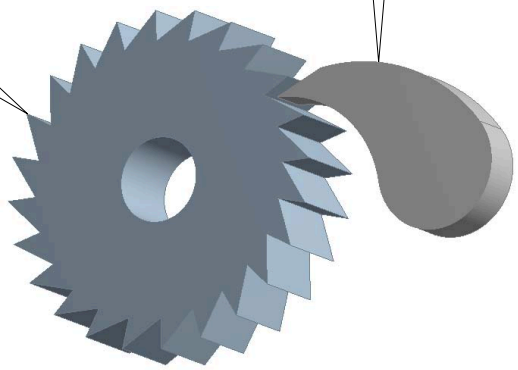
Ratchet Wheel

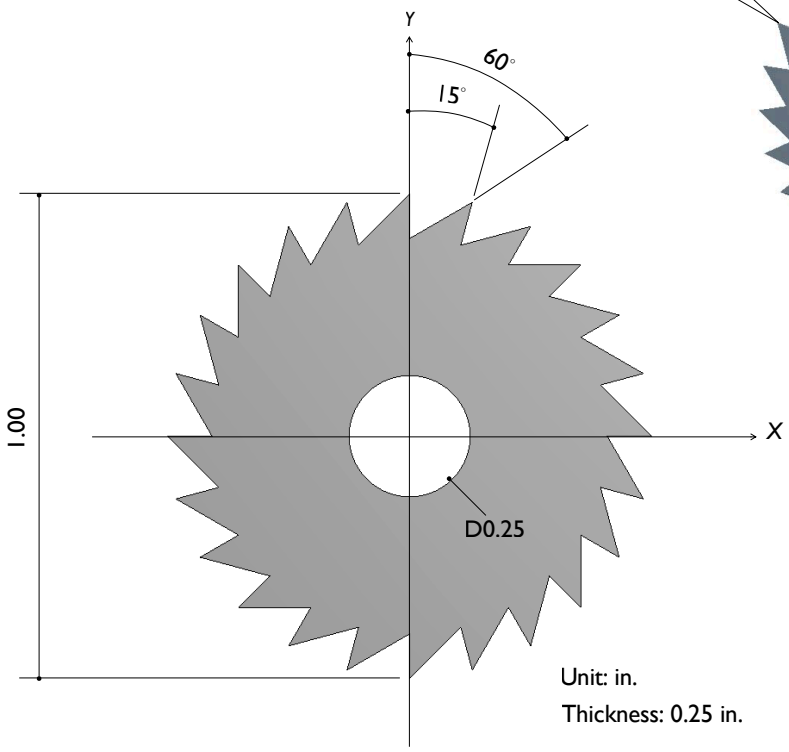


1.2-1 About the Ratchet Wheel

[1] A ratchet wheel rotates in a certain direction controlled by a ratchet stop [2]. In this section, we'll create a 3D model for this ratchet wheel.

[2] The ratchet stop is used to control the rotational direction of the ratchet wheel. The ratchet stop will be created in the next section.





[3] Details of the ratchet wheel.

Unit: in.

Thickness: 0.25 in.

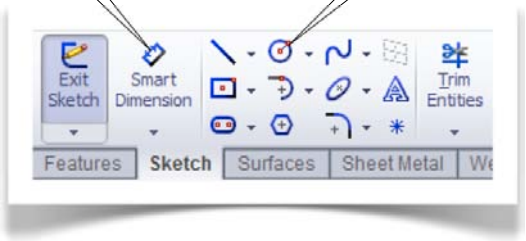
1.2-2 Start Up

[1] Launch **SolidWorks** and create a new part (1.1-2). Set up **IPS** unit system with 2 decimal places for the length unit (1.1-3). Start a sketch on **Front** plane (1.1-4[1, 2]).

1.2-3 Draw a Construction Circle

[3] Select **Smart Dimension**.

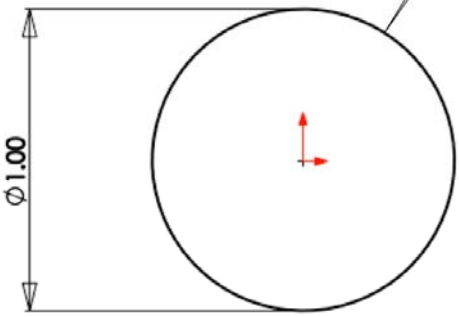
[1] Select **Circle** from the **Sketch Toolbar**.

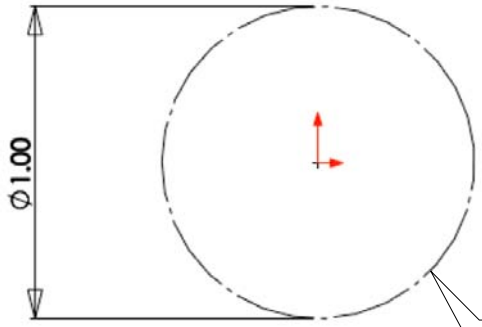


[2] Draw a circle centered at the origin.

[5] Press **ESC** to dismiss **Smart Dimension**. Right-click the circle and select **Construction Geometry** to convert the circle into a construction circle.

[4] Specify the diameter of the circle (1.00 in).

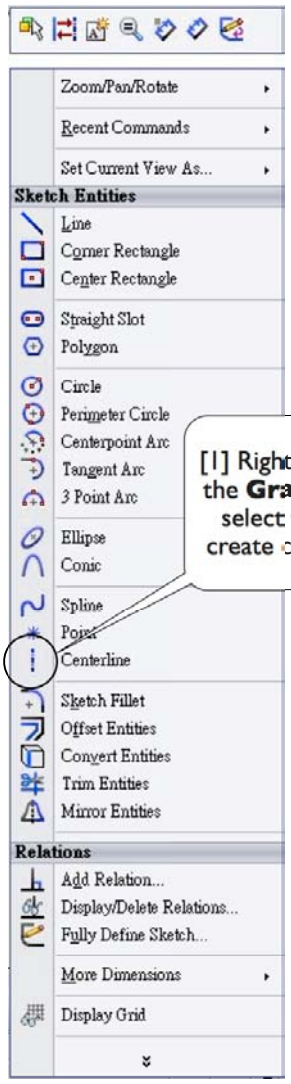




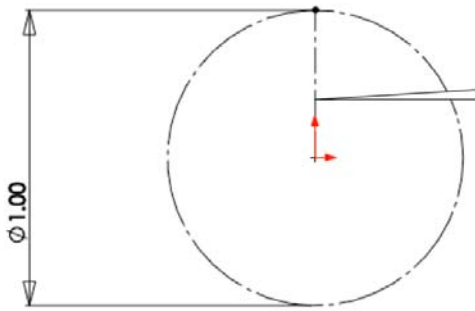
[6] The circle become centerlined. In **SolidWorks**, **Construction Geometry** is represented by centerlines.

[7] **Construction Geometry**
Frequently used **Construction Geometries** include construction lines and construction circles. A construction line can be finite length or infinite length. A **Construction Geometry** is used for reference only, it is not a geometric entity.

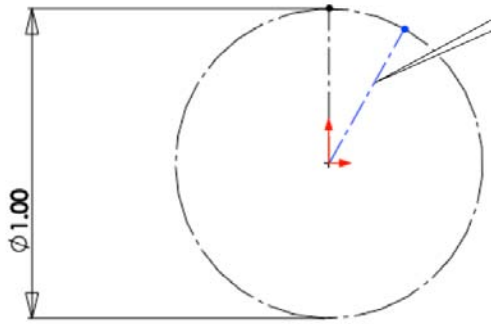
1.2-4 Draw Construction Lines



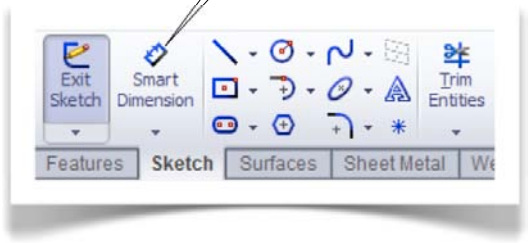
[1] Right-click anywhere in the **Graphics Area** and select **Centerline** to create construction lines.



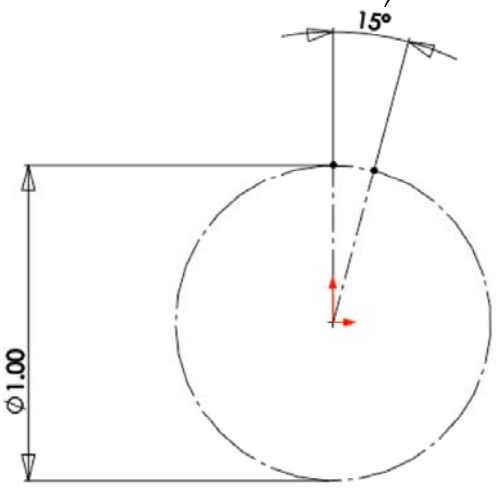
[2] Draw a vertical line from the origin to the upper quarter point of the circle. Double-click anywhere to end the drawing without dismiss the **Centerline** command.



[3] Draw another centerline from the origin to the edge of the circle.

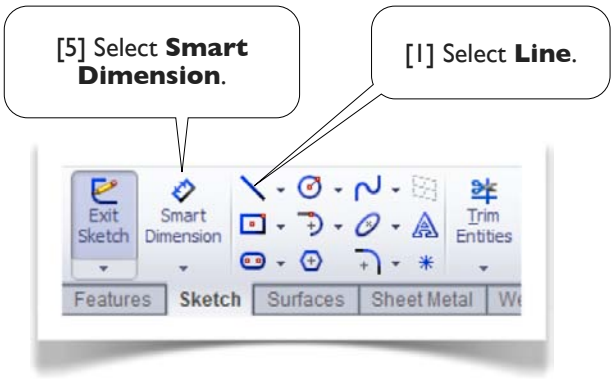


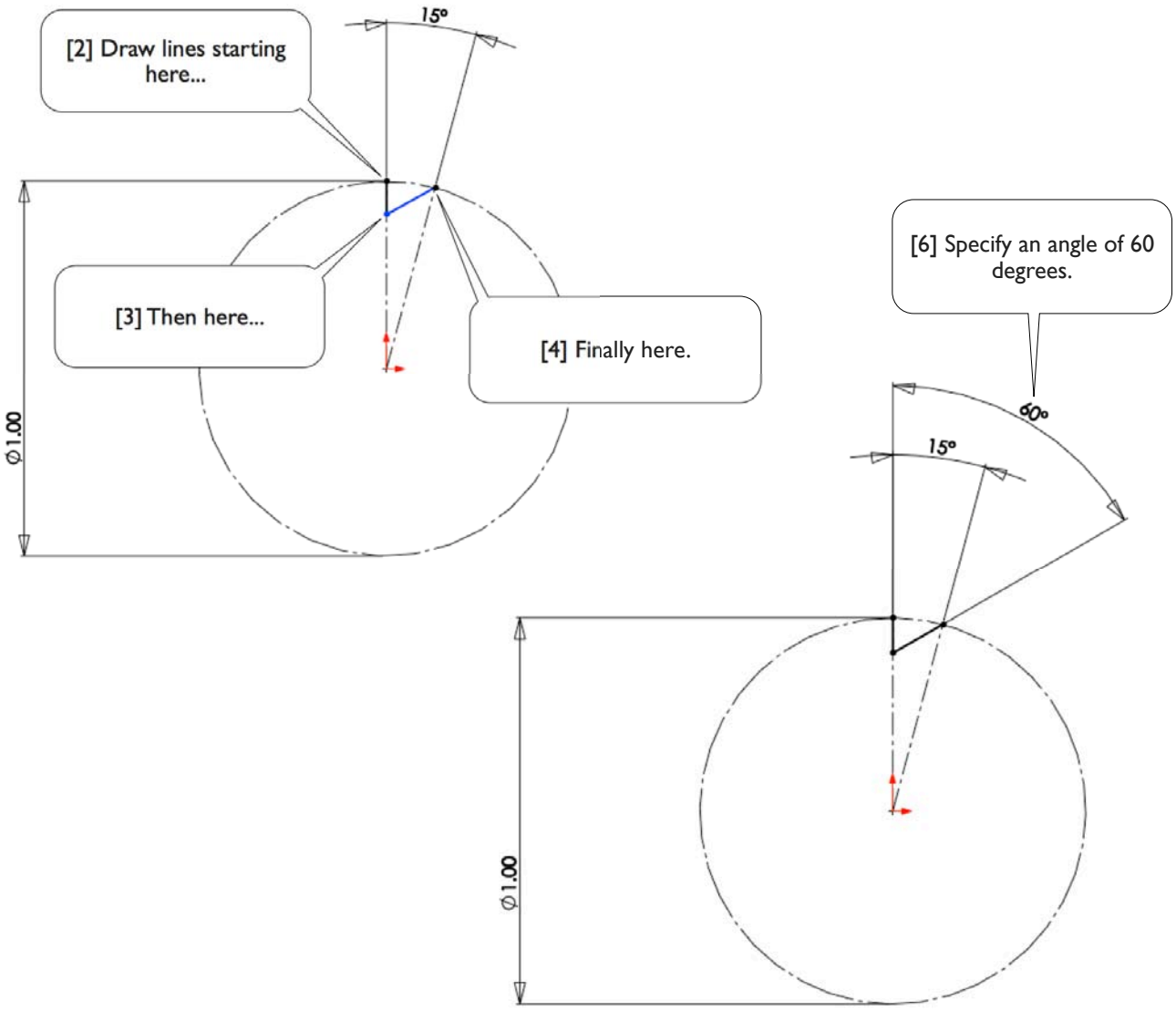
[4] Select **Smart Dimension**.



[5] Click two centerlines one after the other to create an angle dimension; type 15 (degrees).

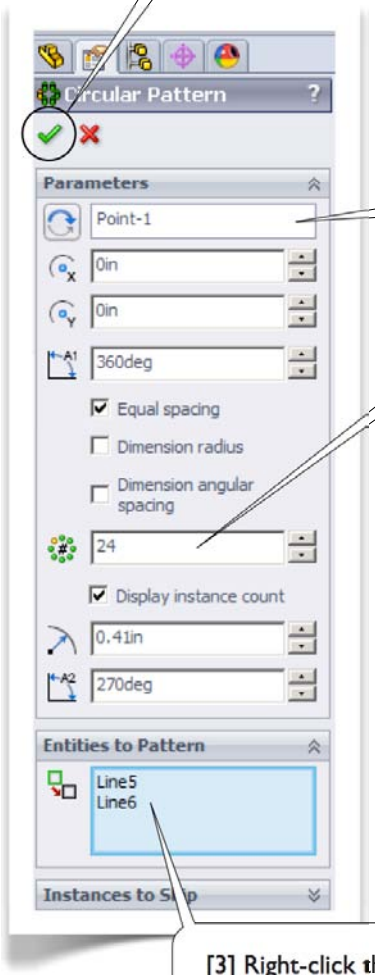
1.2-5 Draw a Tooth





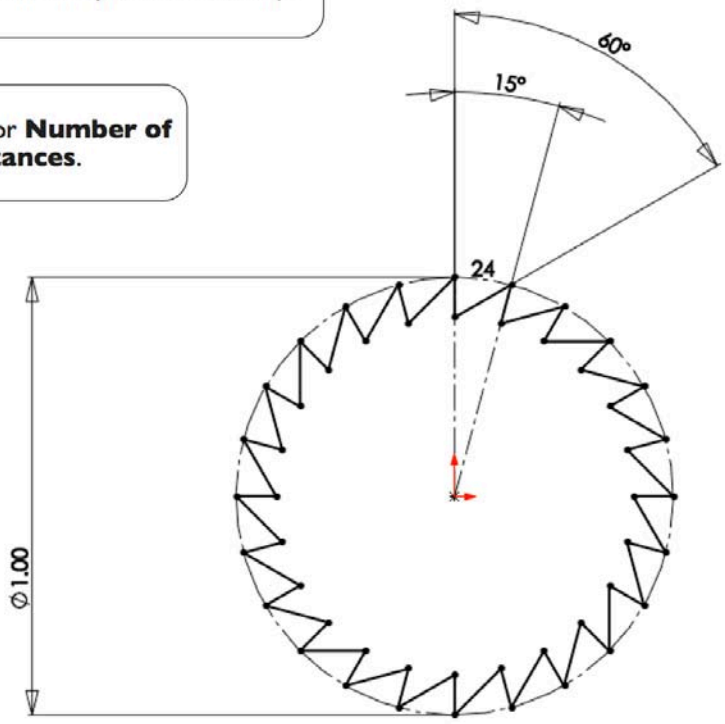
1.2-6 Duplicate the Tooth

[4] Click **OK**.



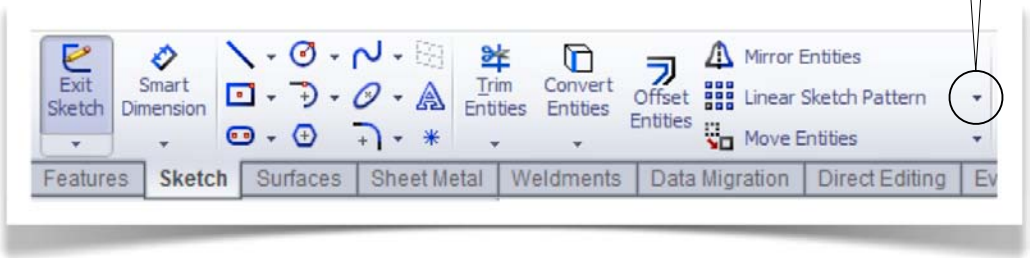
[1] From **Pull-Down Menus**, select **Tools>Sketch Tools>Circular Pattern**. And select the centerlined circle (to define the pattern direction).

[2] Type 24 for **Number of Instances**.

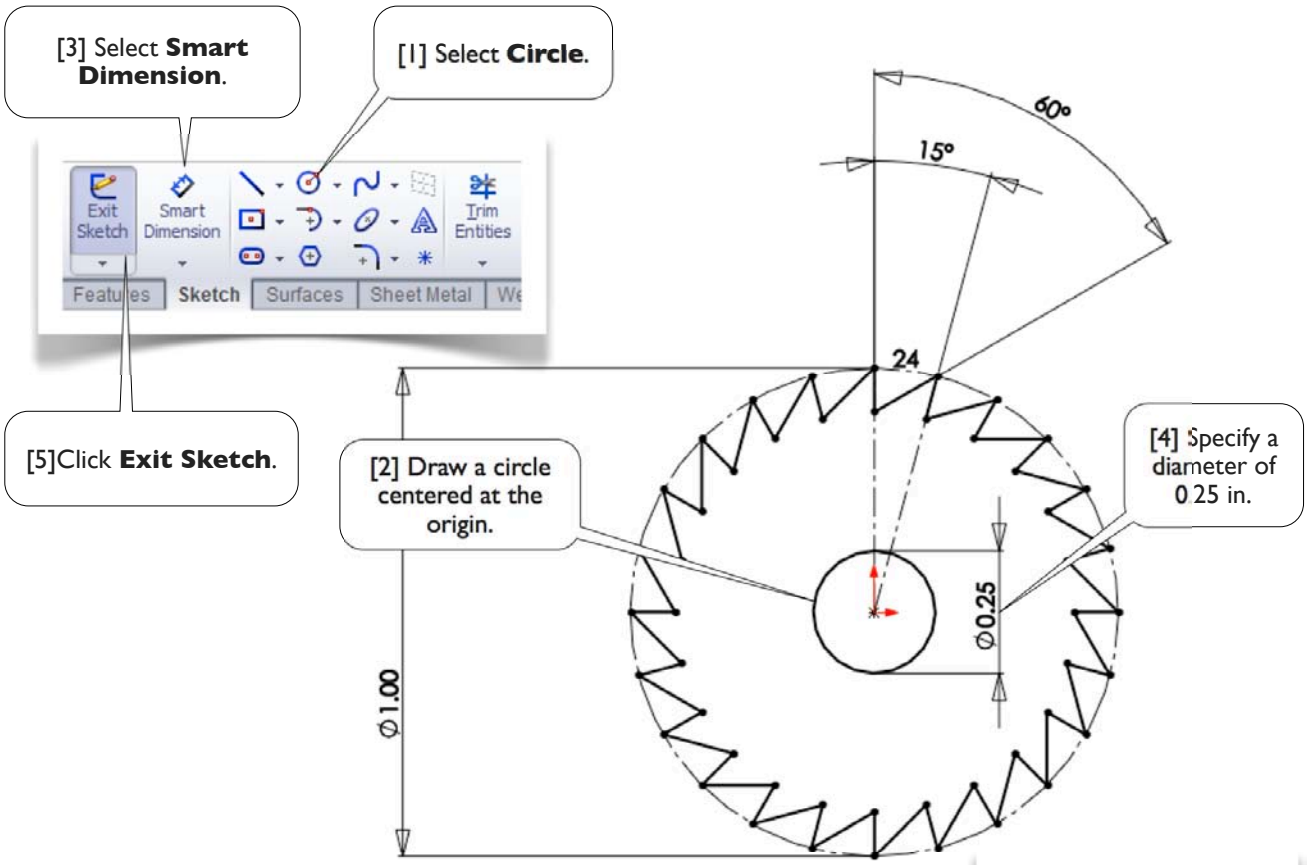


[3] Right-click this box and select **Clear Selections** from the **Context Menu** and then select the two line segments (created in 1.2-5) for **Entities to Pattern**.

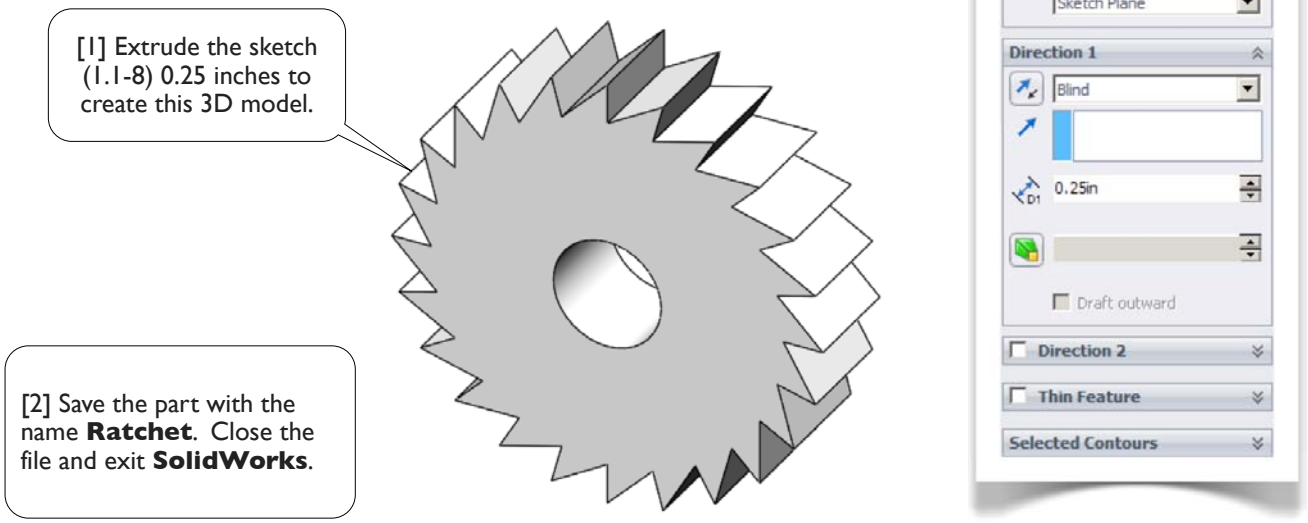
[5] The **Circular Sketch Pattern** command is also available by clicking the arrow next to **Linear Sketch Pattern**.



1.2-7 Draw the Hole

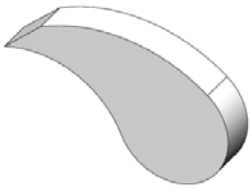


1.2-8 Generate 3D Model

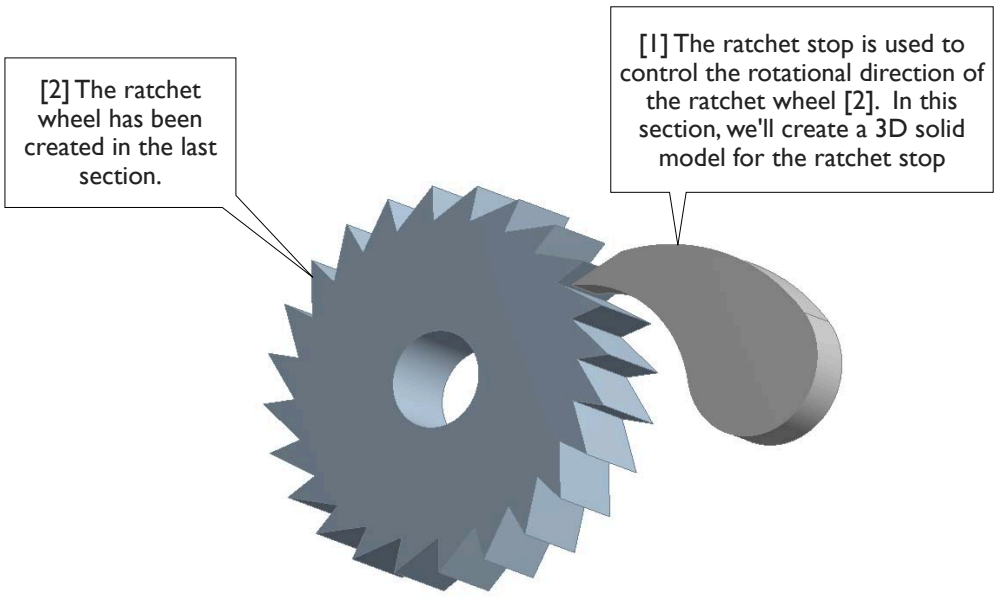


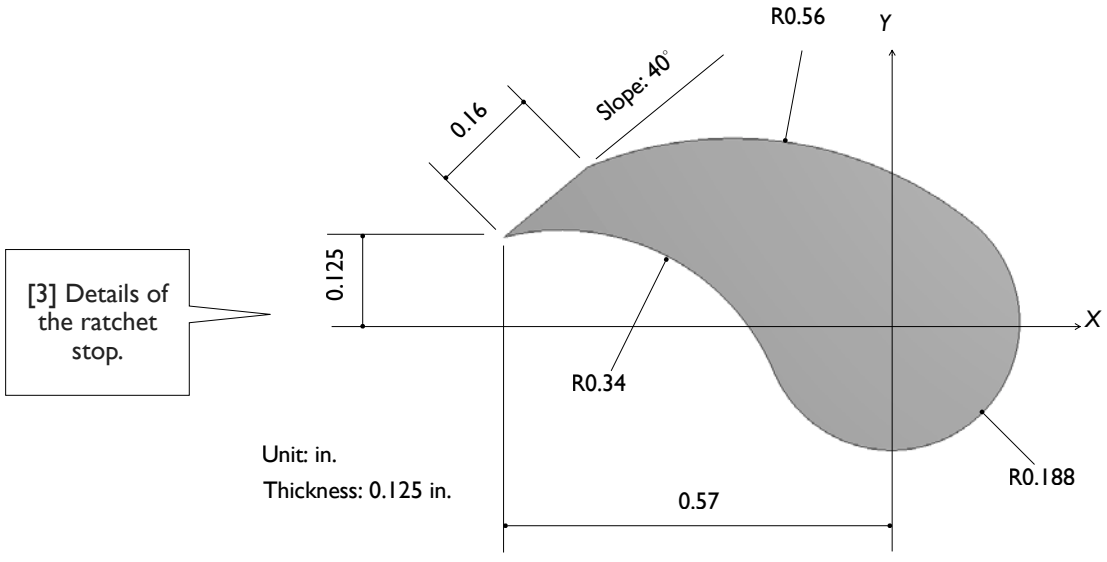
Section 1.3

Ratchet Stop



1.3-1 About the **Ratchet Stop**

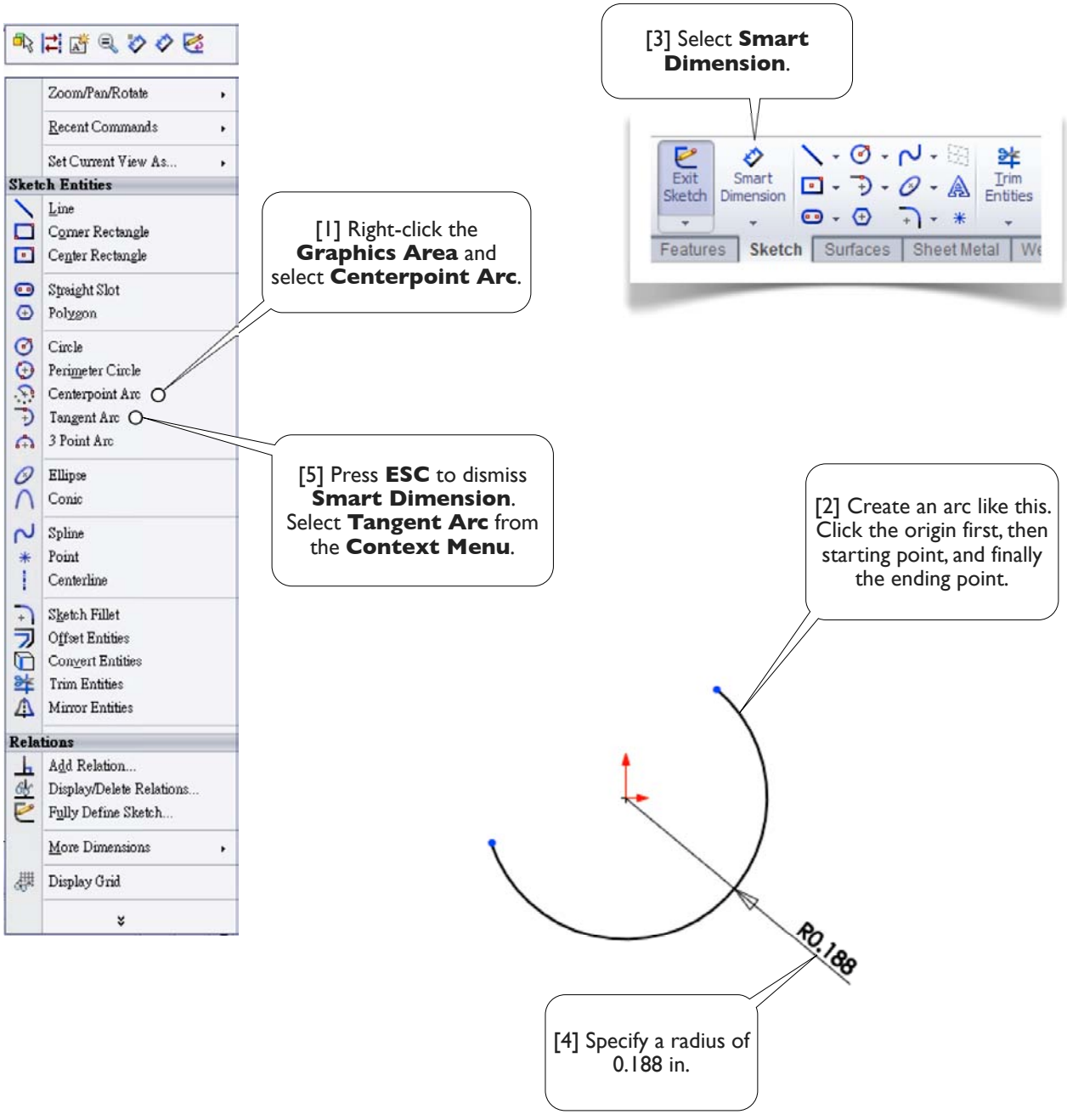


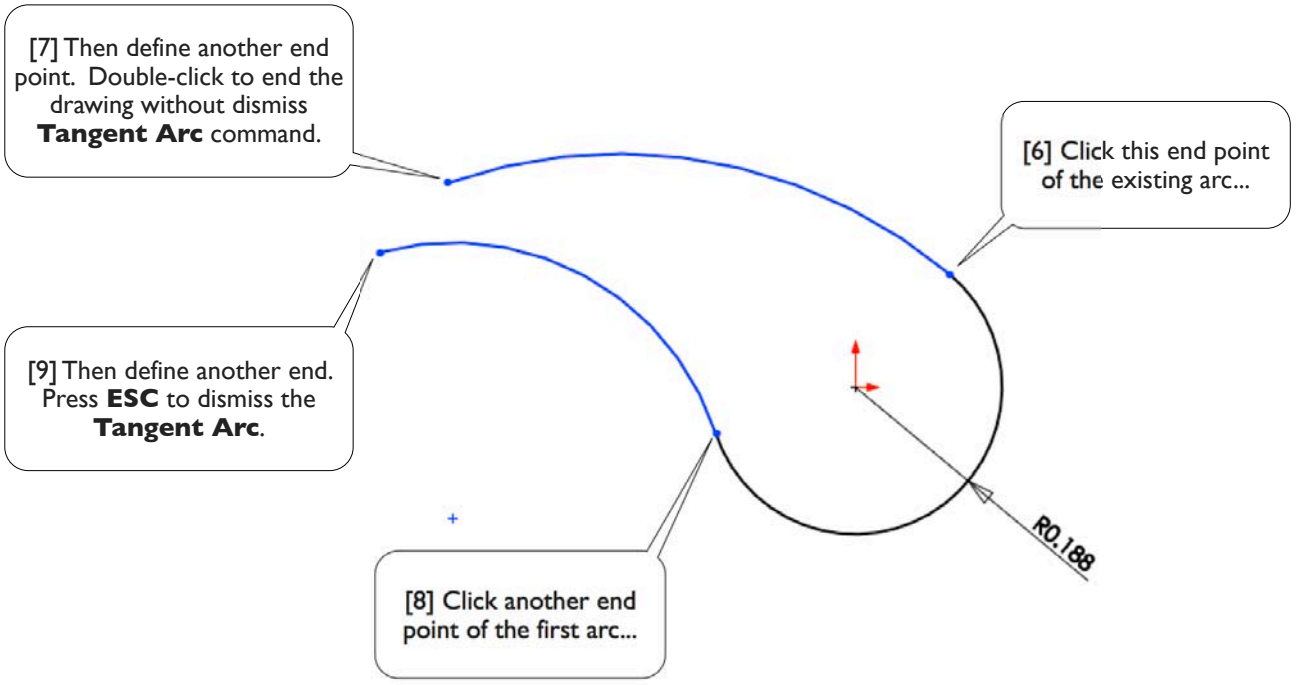


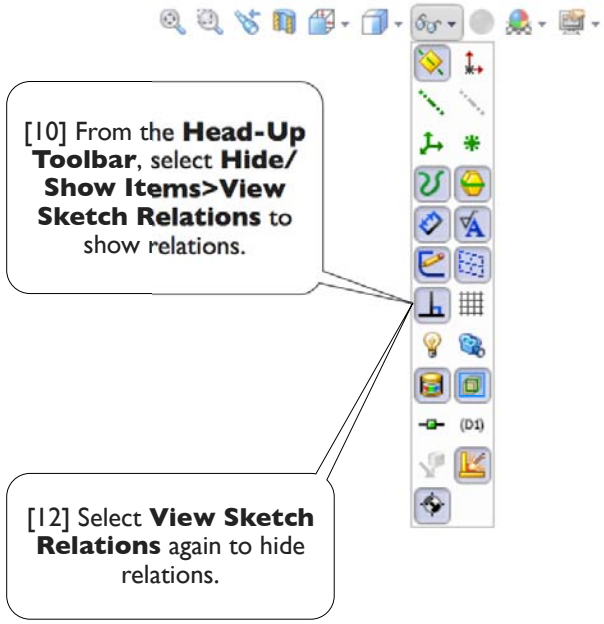
1.3-2 Start Up

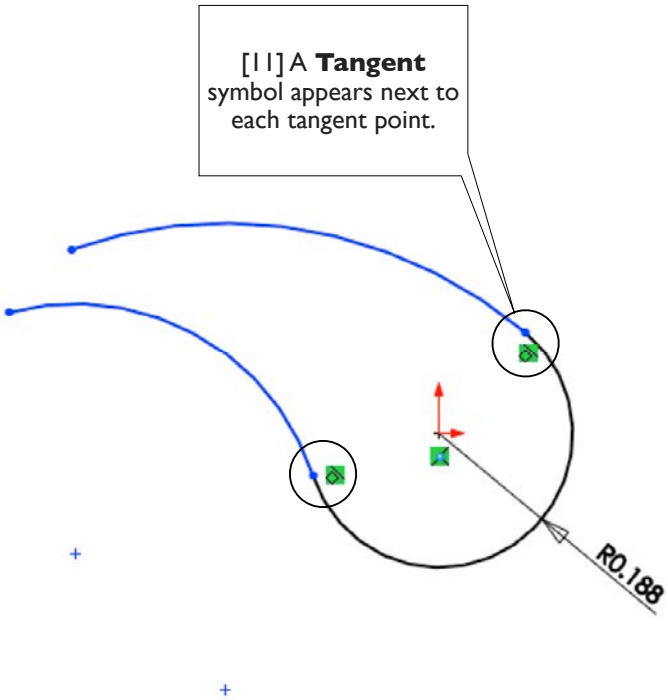
[1] Launch **SolidWorks** and create a new part (1.1-2). Set up **IPS** unit system with 3 decimal places for the length unit (1.1-3). Create a sketch on **Front** plane (1.1-4[1, 2]).

1.3-3 Draw the Sketch

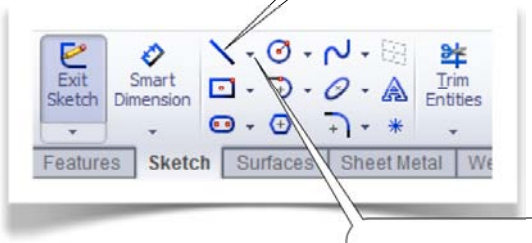



[7] Then define another end point. Double-click to end the drawing without dismiss **Tangent Arc** command.
[6] Click this end point of the existing arc...
[9] Then define another end. Press **ESC** to dismiss the **Tangent Arc**.
[8] Click another end point of the first arc...
R0.188

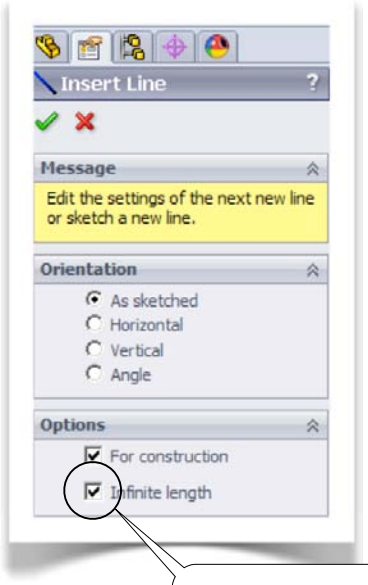

[10] From the **Head-Up Toolbar**, select **Hide/ Show Items>View Sketch Relations** to show relations.
[12] Select **View Sketch Relations** again to hide relations.


[11] A **Tangent** symbol appears next to each tangent point.
R0.188

[13] Select **Line** command.



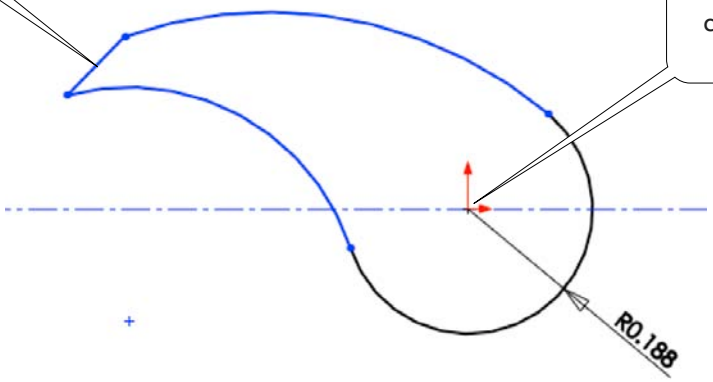
[15] Click the arrow head next to **Line** command and select **Centerline**.



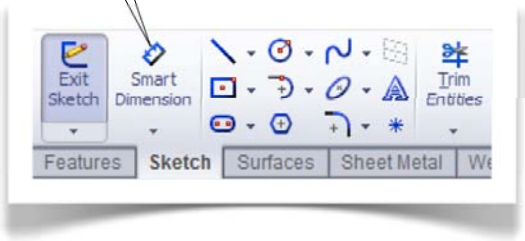
[16] In the **Property Box**, select **Infinite length**.

[14] Draw this line.

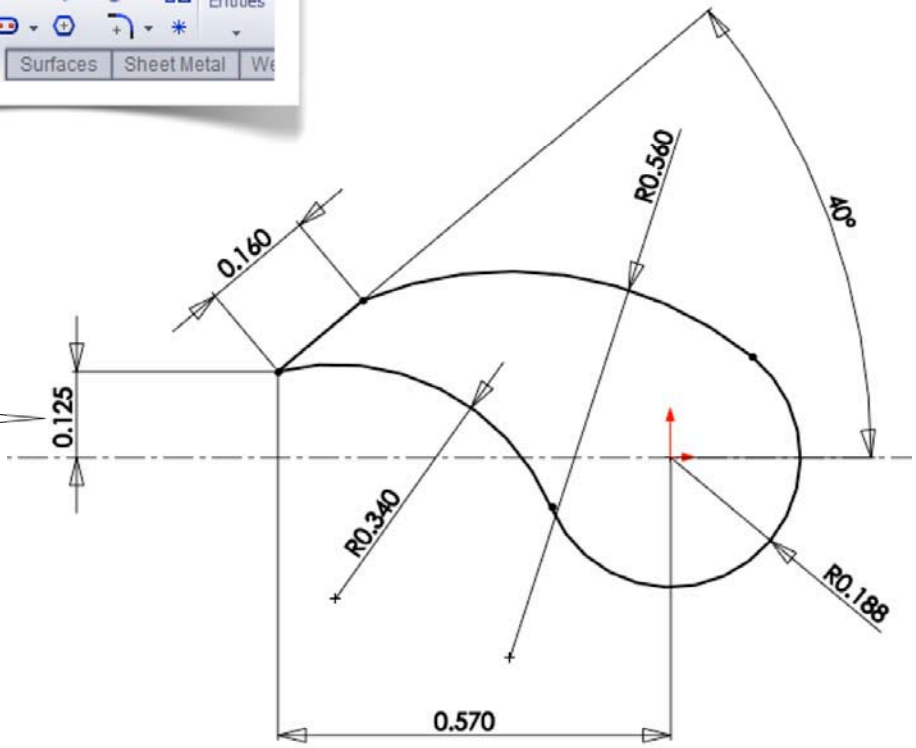
[17] Click the origin and then click any horizontal point to create a construction line of infinite length.



[18] Select **Smart Dimension**.

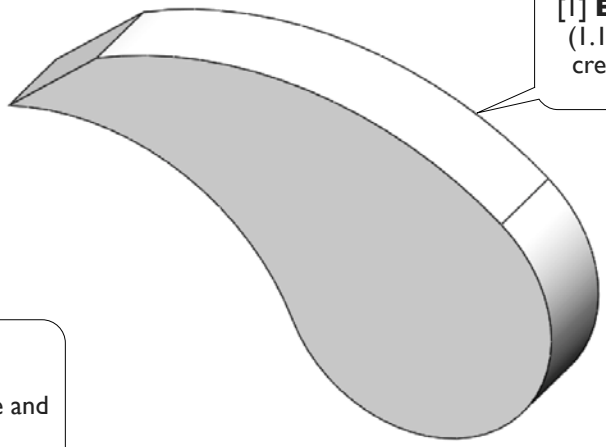


[19] Finish up the sketch by specify the rest of the dimensions. All entities must be black-colored.



1.3-4 Generate 3D Model

[1] **Extrude** the sketch (1.1-8) 0.125 inches to create this 3D model.



[2] Save the part with the name **Stop**. Close the file and exit **SolidWorks**.