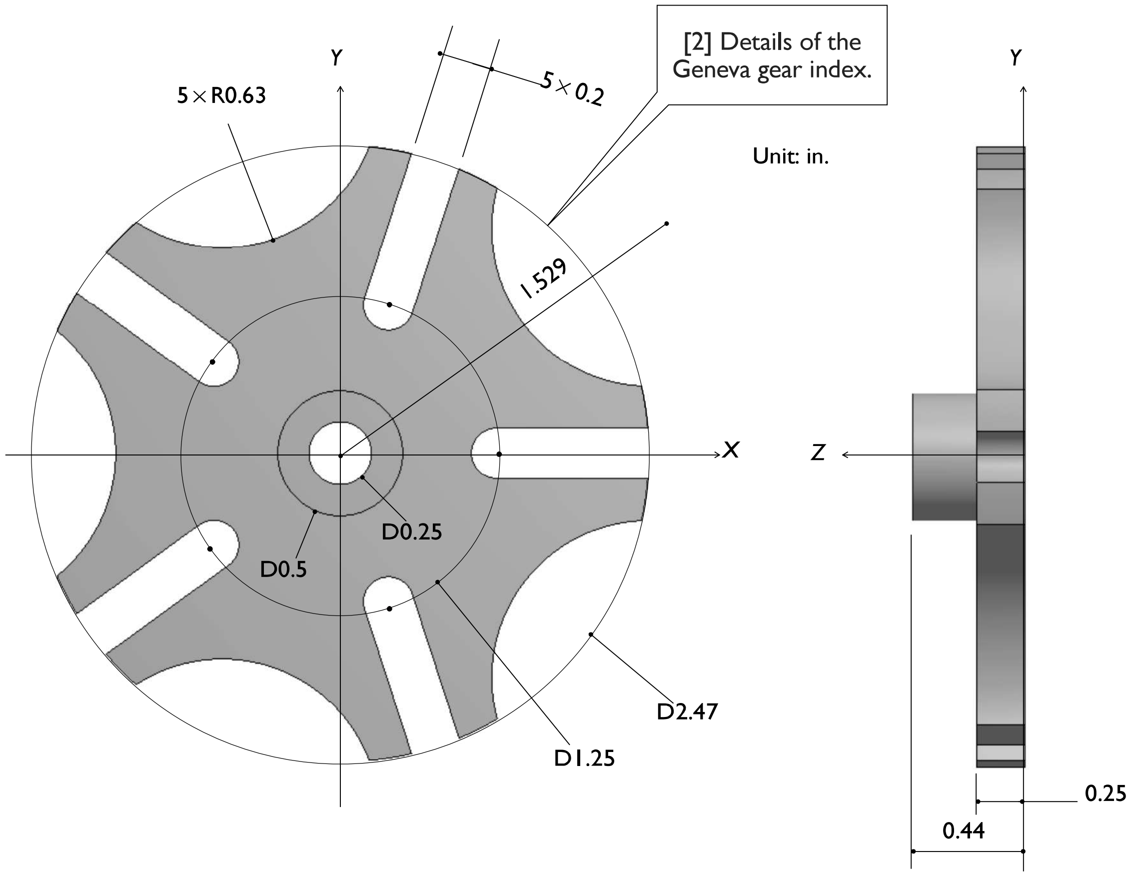
Section 2.2

Geneva Gear Index

2.2-1 About the Geneva Gear Index

[1] In this exercise, we'll create a 3D solid model for a Geneva gear index [2].

[2] Details of the Geneva gear index.

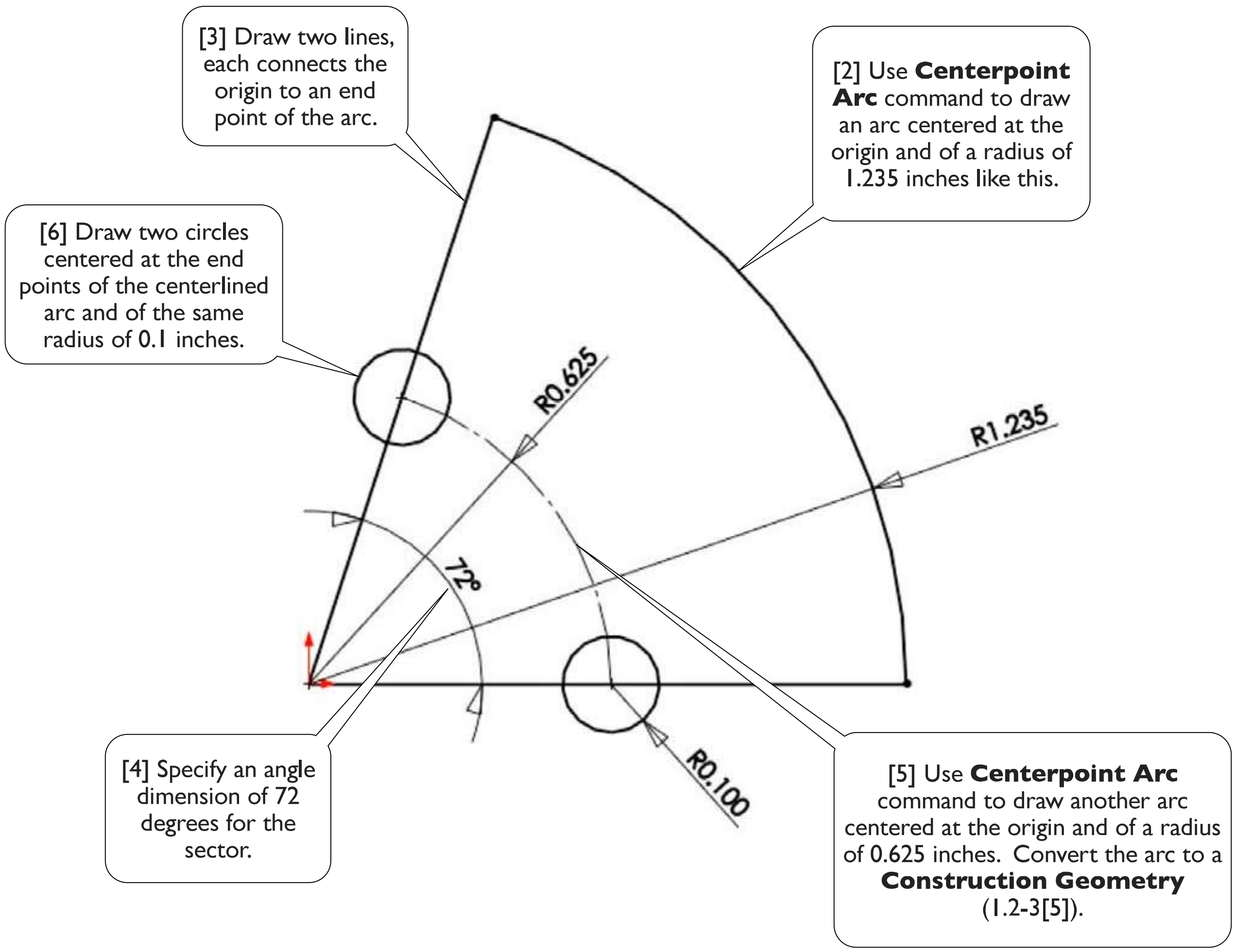


2.2-2 Start Up

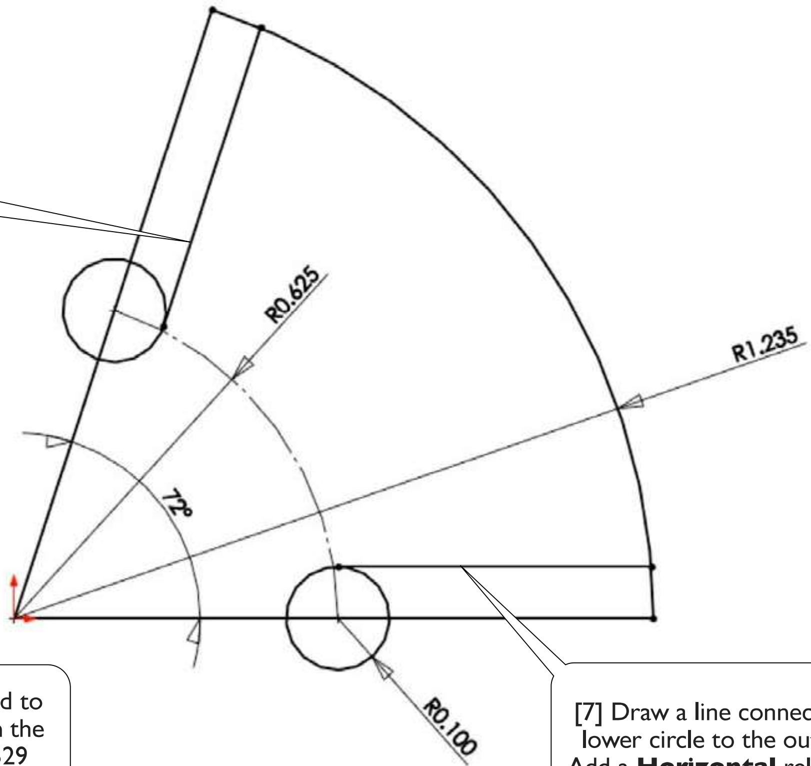
[1] Launch **SolidWorks** and create a new part. Set up **IPS** unit system with 3 decimal places for the length unit.

2.2-3 Draw a Sketch for 1/5 of the Gear Index

[1] Create a sketch on **Front** plane.

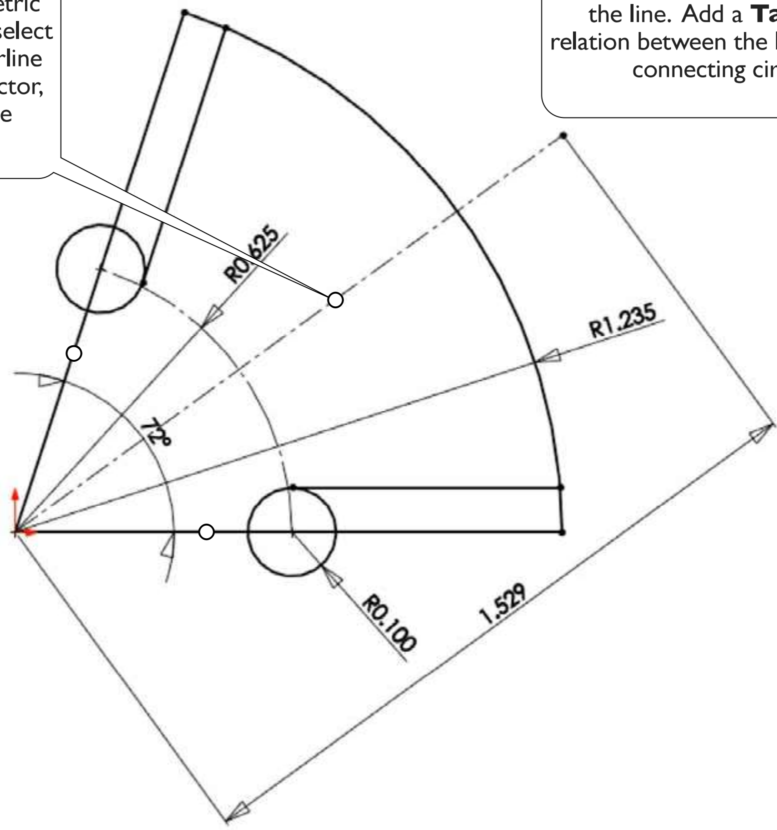


[8] Draw a line connecting the upper circle to the outer arc. Add a **Parallel** relation between the line and the line next to it. Add a **Tangent** relation between the line and the connecting circle.

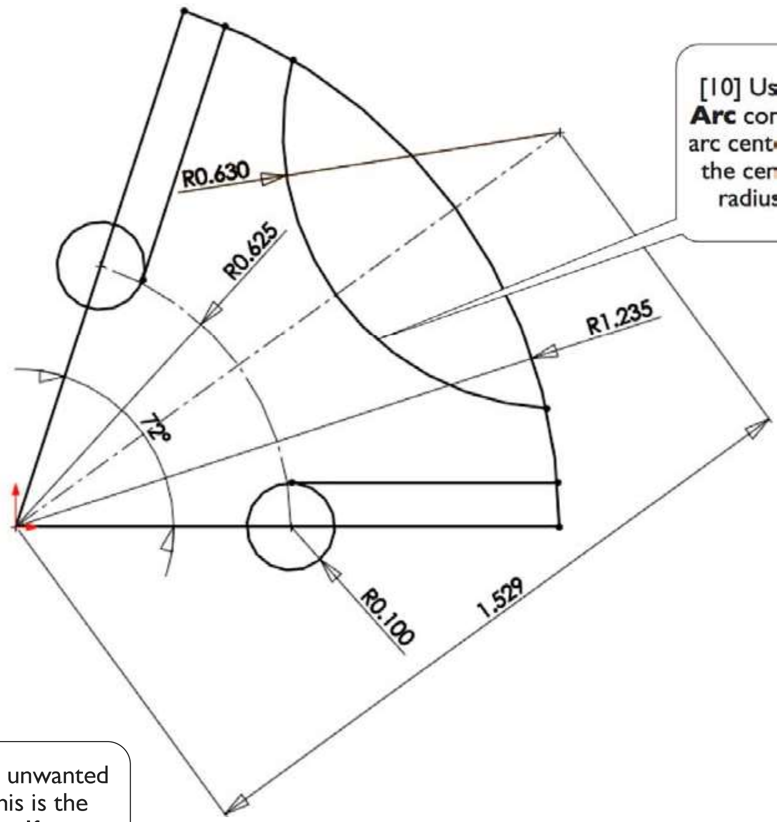


[7] Draw a line connecting the lower circle to the outer arc. Add a **Horizontal** relation on the line. Add a **Tangent** relation between the line and the connecting circle.

[9] Use **Centerline** command to draw a centerline starting from the origin. Specify the length (1.529 inches). Make the sector symmetric about the centerline. To do this, select **Add Relation**, click the centerline and the two edge lines of the sector, and select **Symmetric** in the **Property Box**.



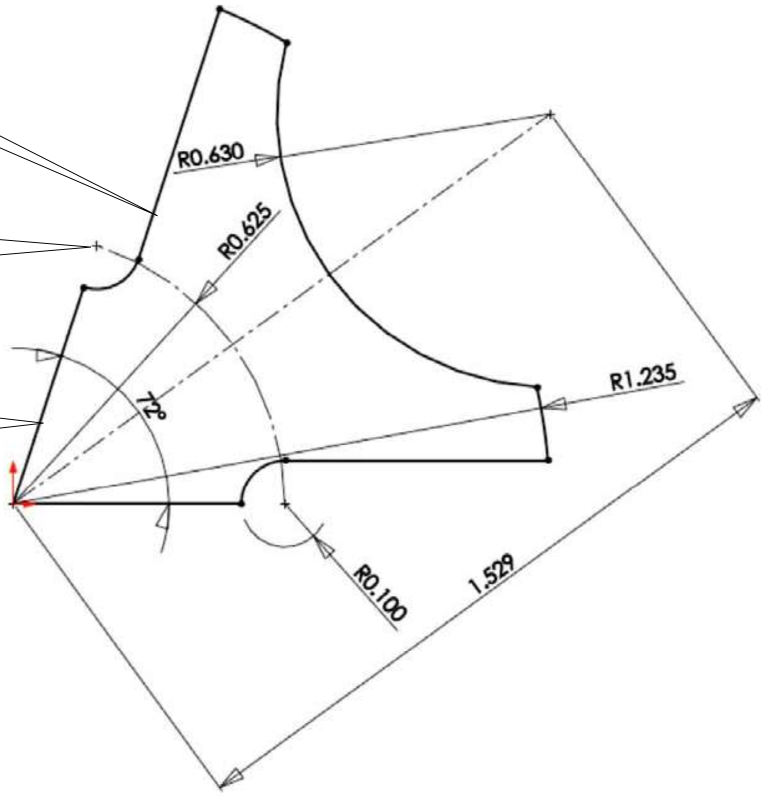
[10] Use **Centerpoint Arc** command to draw an arc centered at one end of the centerline. Specify a radius of 0.63 inches.



[11] Trim away unwanted segments. This is the finished sketch. If some entities turn to blue color, add relations to fix them (see [12-14]).

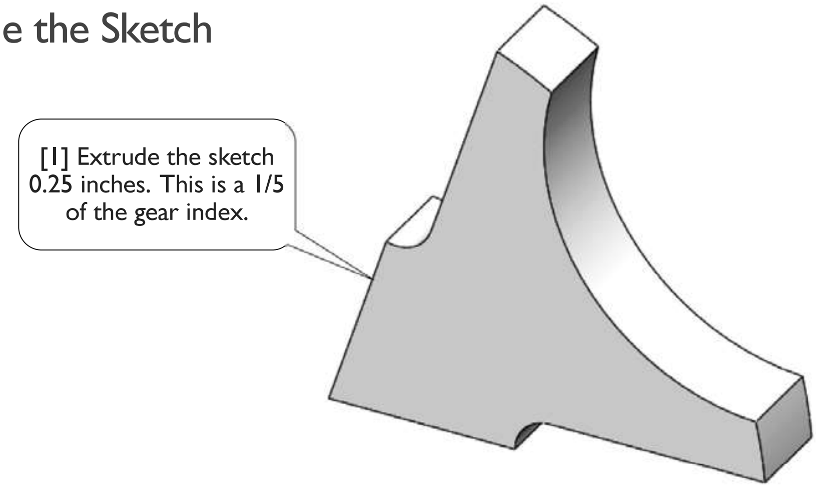
[12] If some entities turn to blue, try to fix them by adding a **Coincident** relation for this point...

[13] And this line. Add another **Coincident** for the other side of the sector.

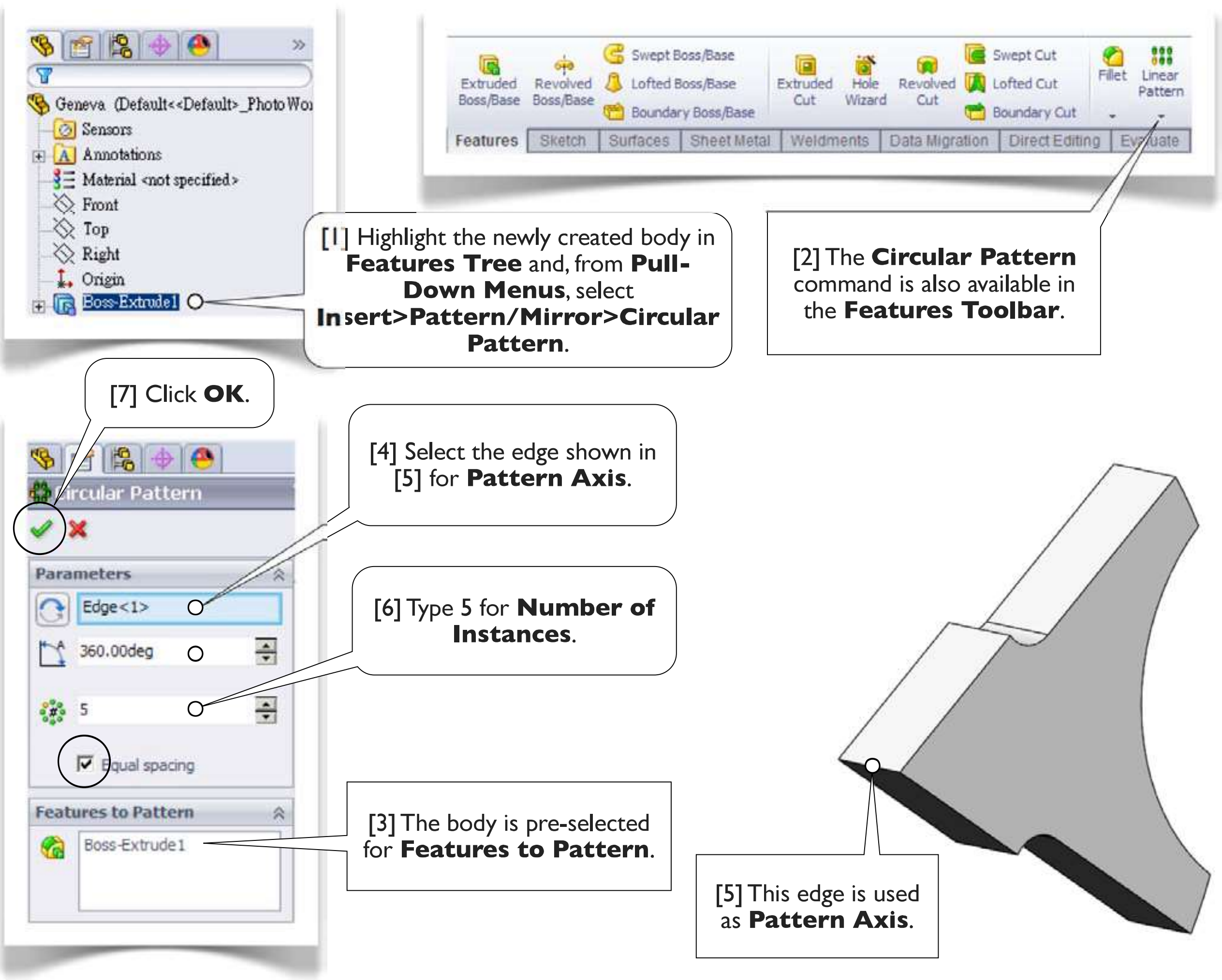


[14] If steps [12, 13] don't fix all the entities. Try to drag an unfixed entity to see what relations should be added.

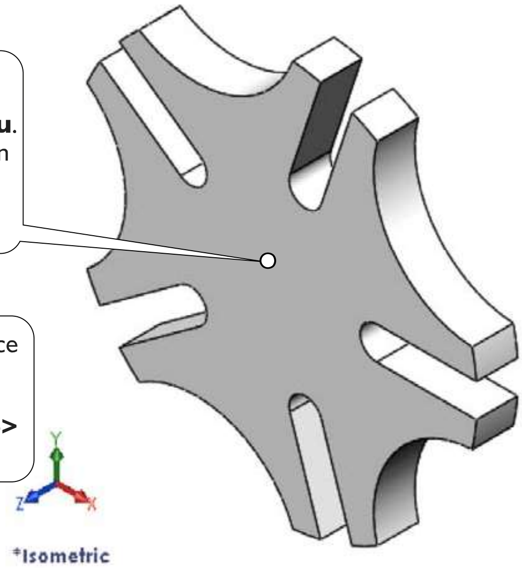
2.2-4 Extrude the Sketch



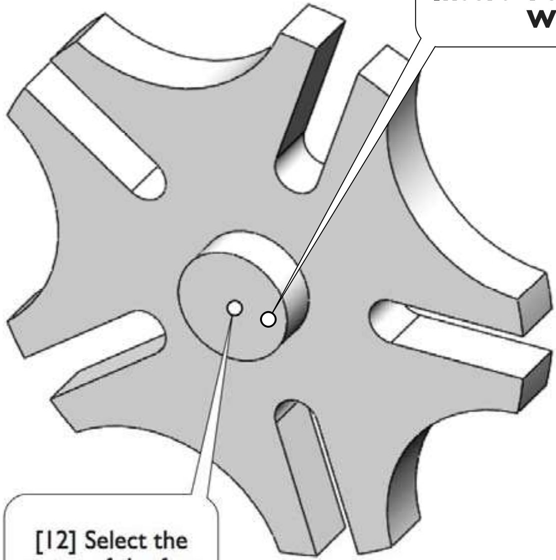
2.2-5 Complete the Full Model



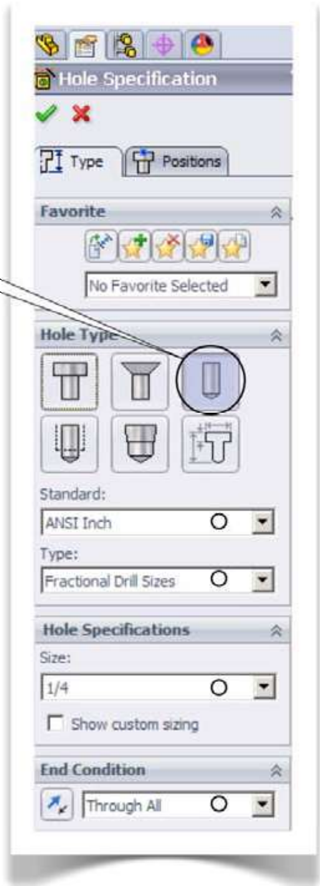
[8] Right-click this face and select **Sketch** from the **Context Menu**. Draw a circle centered at the origin with a diameter of 0.5 inches. Extrude the sketch 0.19 inches.



[9] Click to highlight this face and, from **Pull-Down Menus**, select **Insert>Features>Hole> Wizard...**



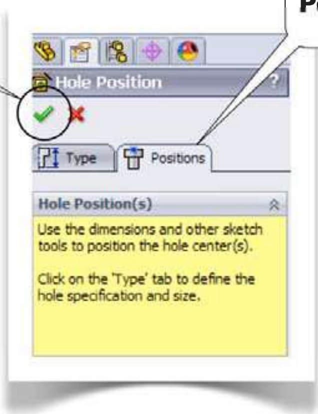
[10] Select **Hole** for **Hole Type** and set up other settings as shown.

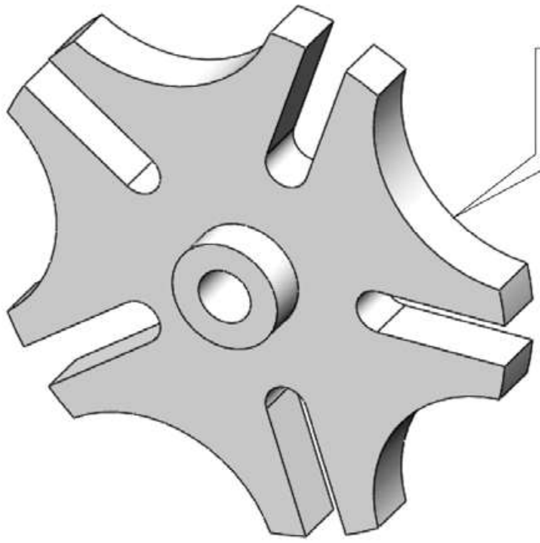


[12] Select the center of the face.

[11] Select **Positions** tab.

[13] Click **OK**.





[14] The finished 3D model.

[15] The **Hole Wizard** command is also available in **Features Toolbar**.

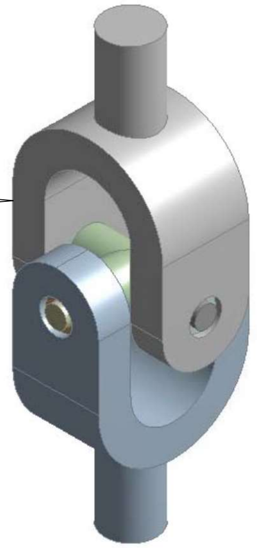
[16] Save the part with the file name **Geneva**. Close the file and exit **SolidWorks**.

Section 2.3

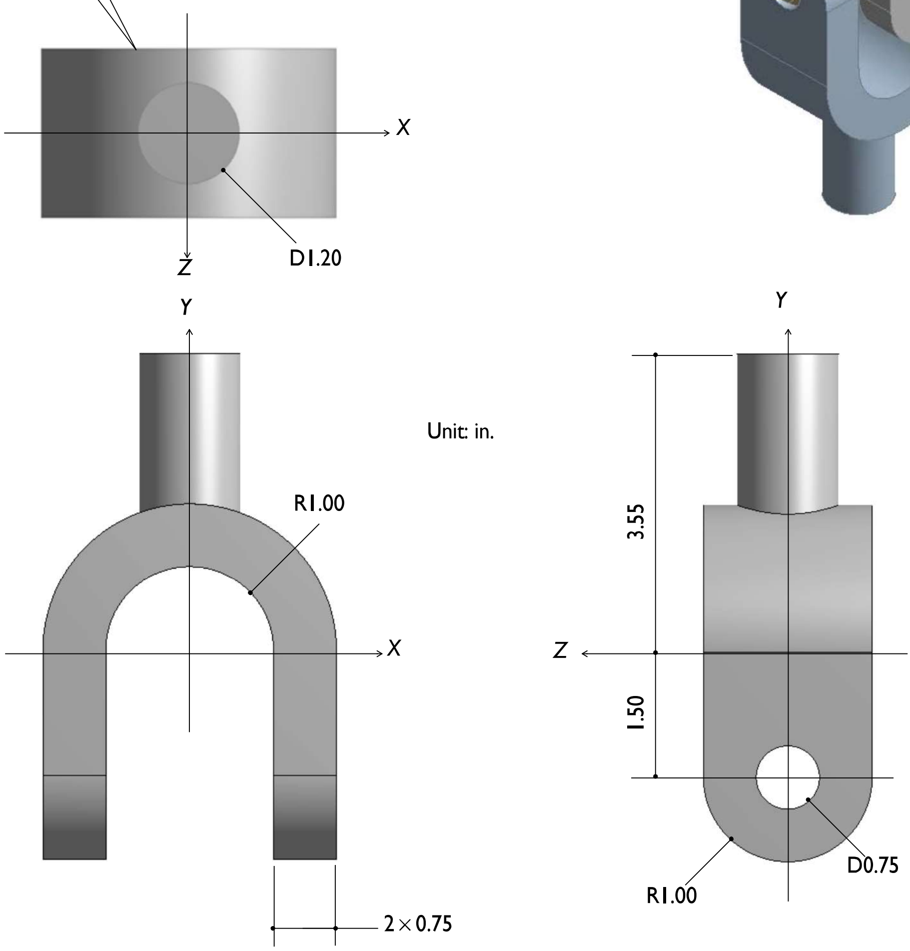
Yoke

2.3-1 About the Yoke

[1] The yoke is a part of a universal joint. In this exercise, we'll create a 3D solid model for the yoke.



[2] Details of the yoke.



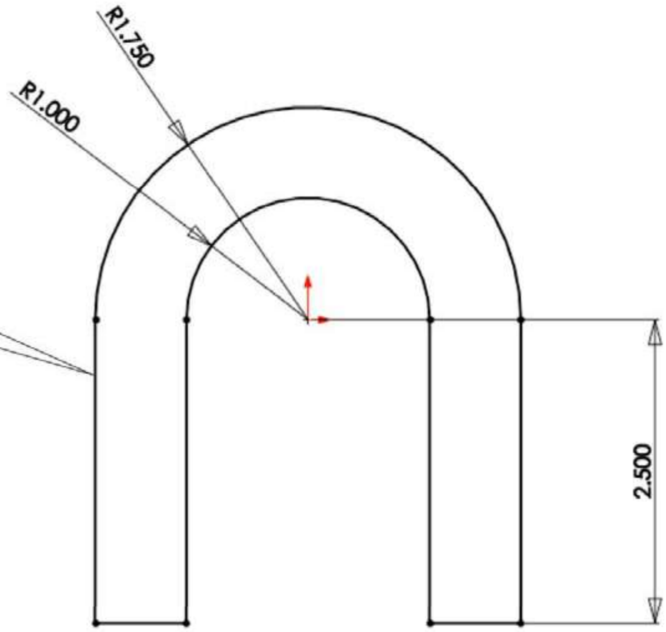
2.3-2 Start Up

[1] Launch **SolidWorks** and create a new part. Set up **IPS** unit system with 3 decimal places for the length unit.

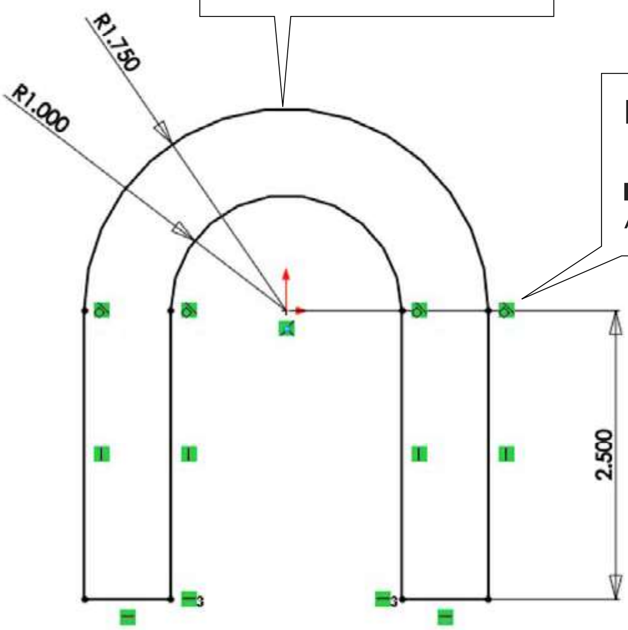
2.3-3 Create a Base Body

[1] Create a sketch on **Front** plane.

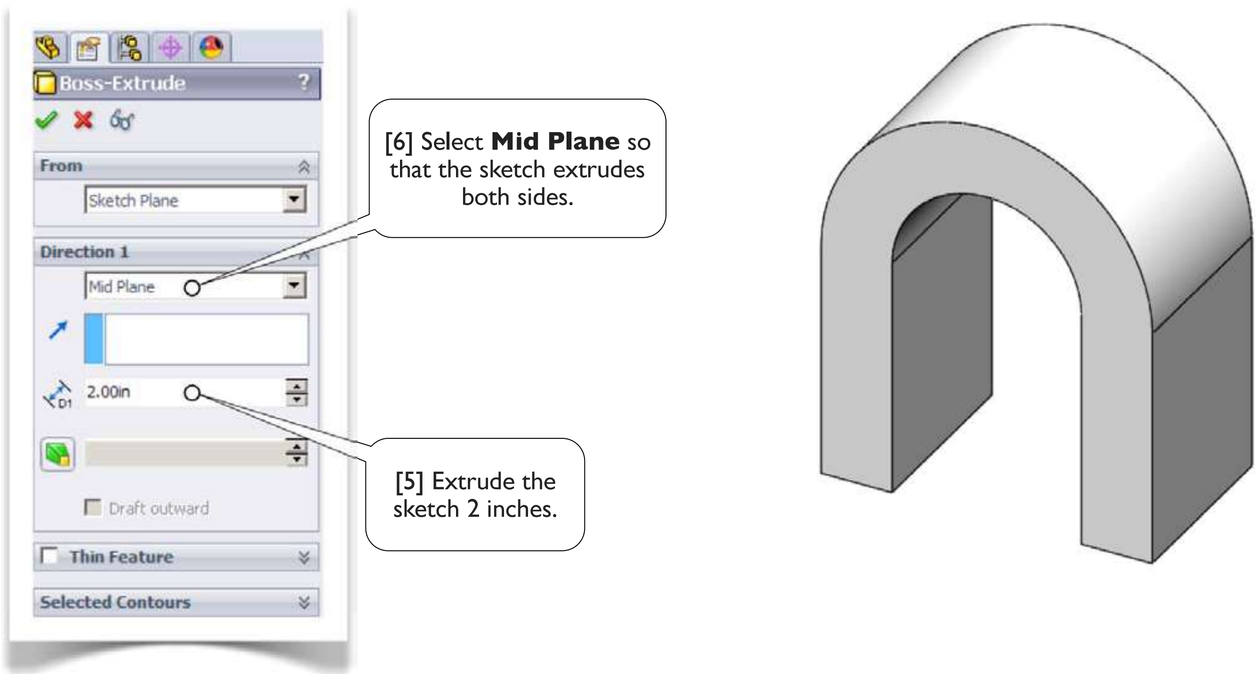
[2] Draw a sketch like this. If there are any blue entities (not well-defined), see [3, 4].



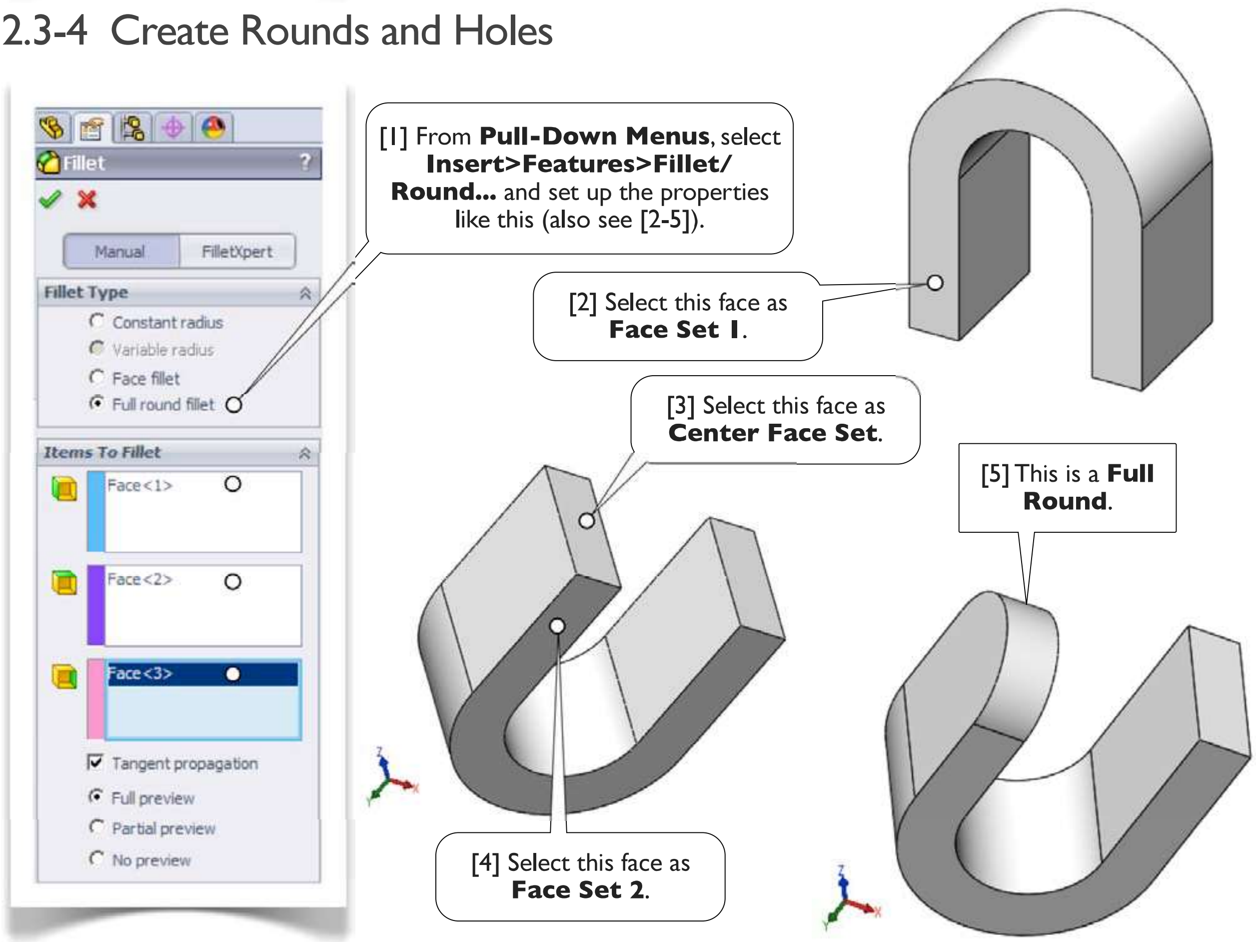
[4] Another way is to drag an unfixed entity to see what relations should be added.



[3] If there are any blue entities, select **Hide/Show Items>View Sketch Relations** to view all relations. Add relations to fix the entities.

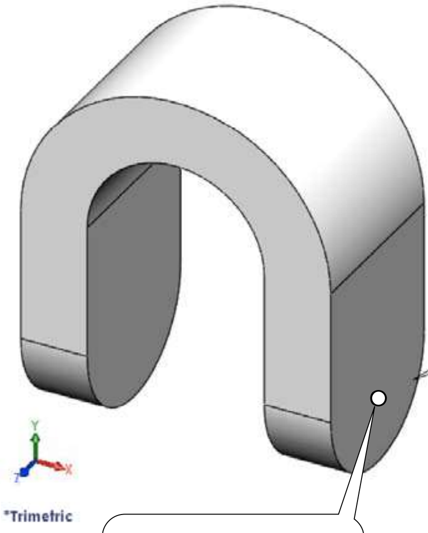


2.3-4 Create Rounds and Holes



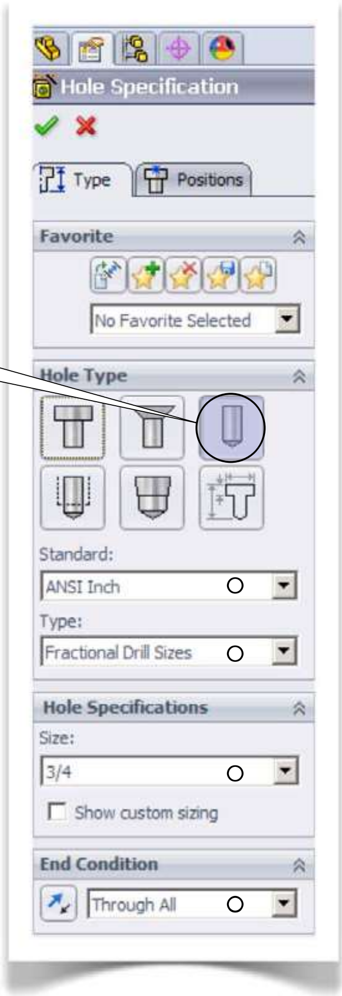
[9] Click **Hole Wizard**.

[6] The **Fillet** command is also available on the **Features Toolbar**. Click it.



[7] Create another **Full Round**.

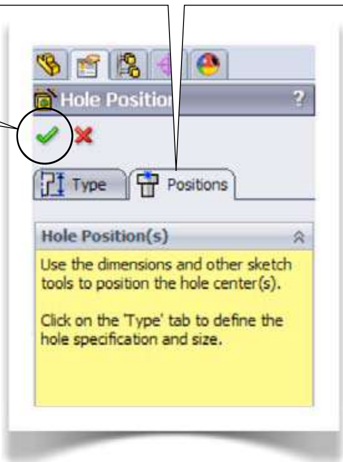
[10] Set up the properties like this.

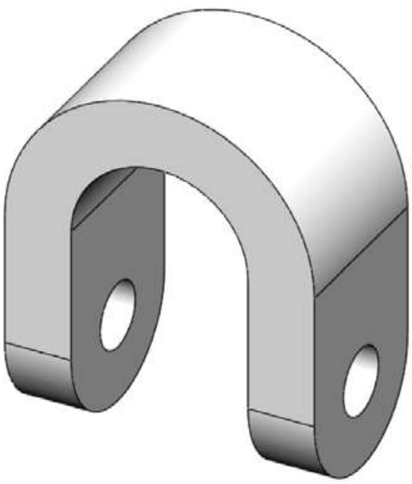


[8] Select this face.

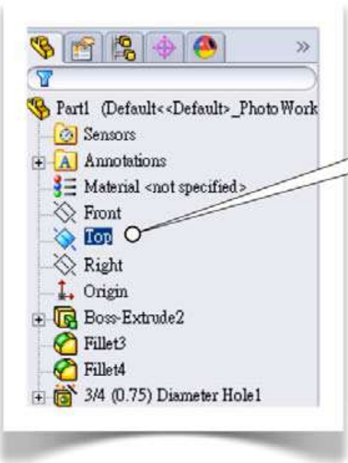
[11] Select **Positions** tab and click the arc-center of the selected face [8]. To locate the arc-center, move your mouse over the arc, the center will show up.

[12] Click **OK**.

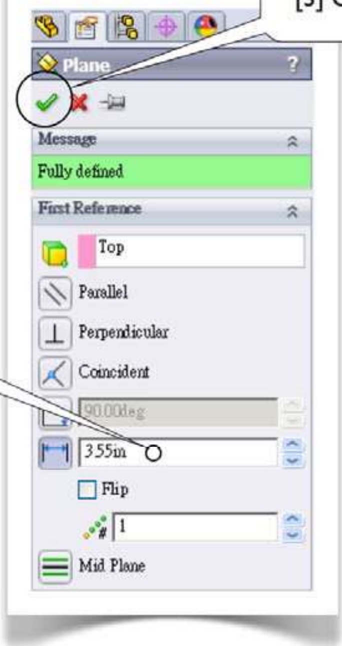




2.3-5 Create a Plane

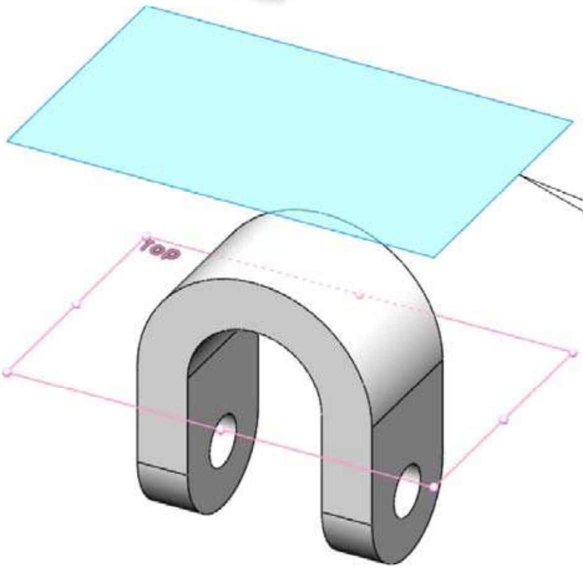


[1] Highlight **Top** and, from **Pull-Down Menus**, select **Insert>Reference Geometry>Plane...**

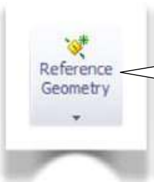


[3] Click **OK**.

[2] Type 3.55 (in) for **Offset distance**.

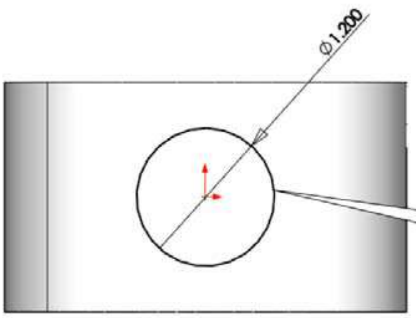


[4] A new plane is created by offsetting **Top** plane 3.55 inches upward.

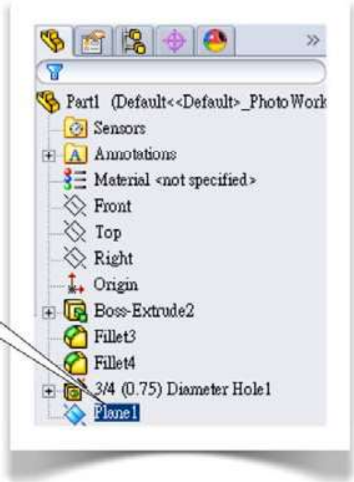


[5] The **Reference Geometry** is also available on the **Features Toolbar**.

2.3-6 Create the Shaft



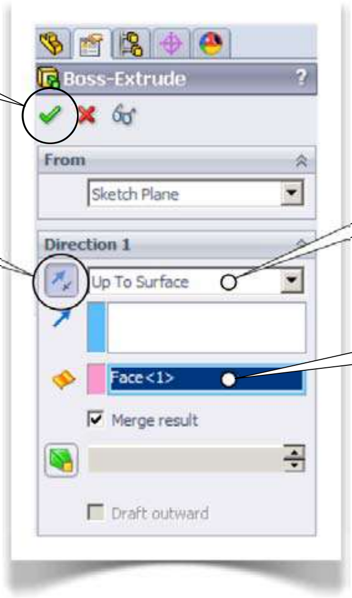
[1] Right-click the newly created plane and select **Sketch** from the **Context Menu**.



[2] Draw a circle centered at the origin and with a diameter of 1.2 inches.

[7] Click **OK**.

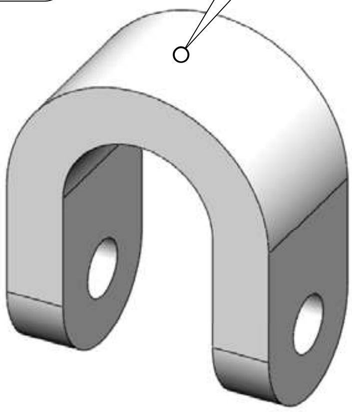
[4] Click **Reverse Direction**.



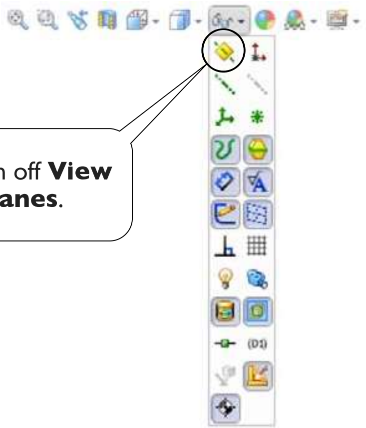
[3] From the **Features Toolbar**, select **Extruded Boss/Base** and select **Up To Surface** for **End Condition**.

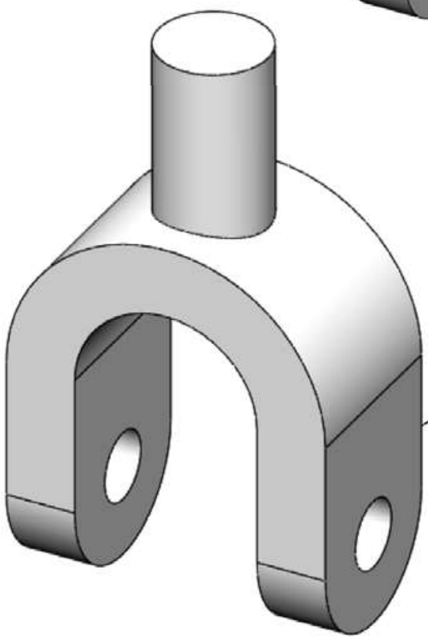
[5] Select the top face of the existing body (see [6]).

[6] Face<1>.



[8] Turn off **View Planes**.



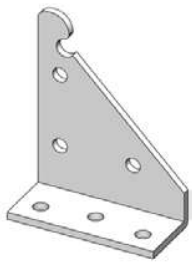


[9] The finished 3D model.

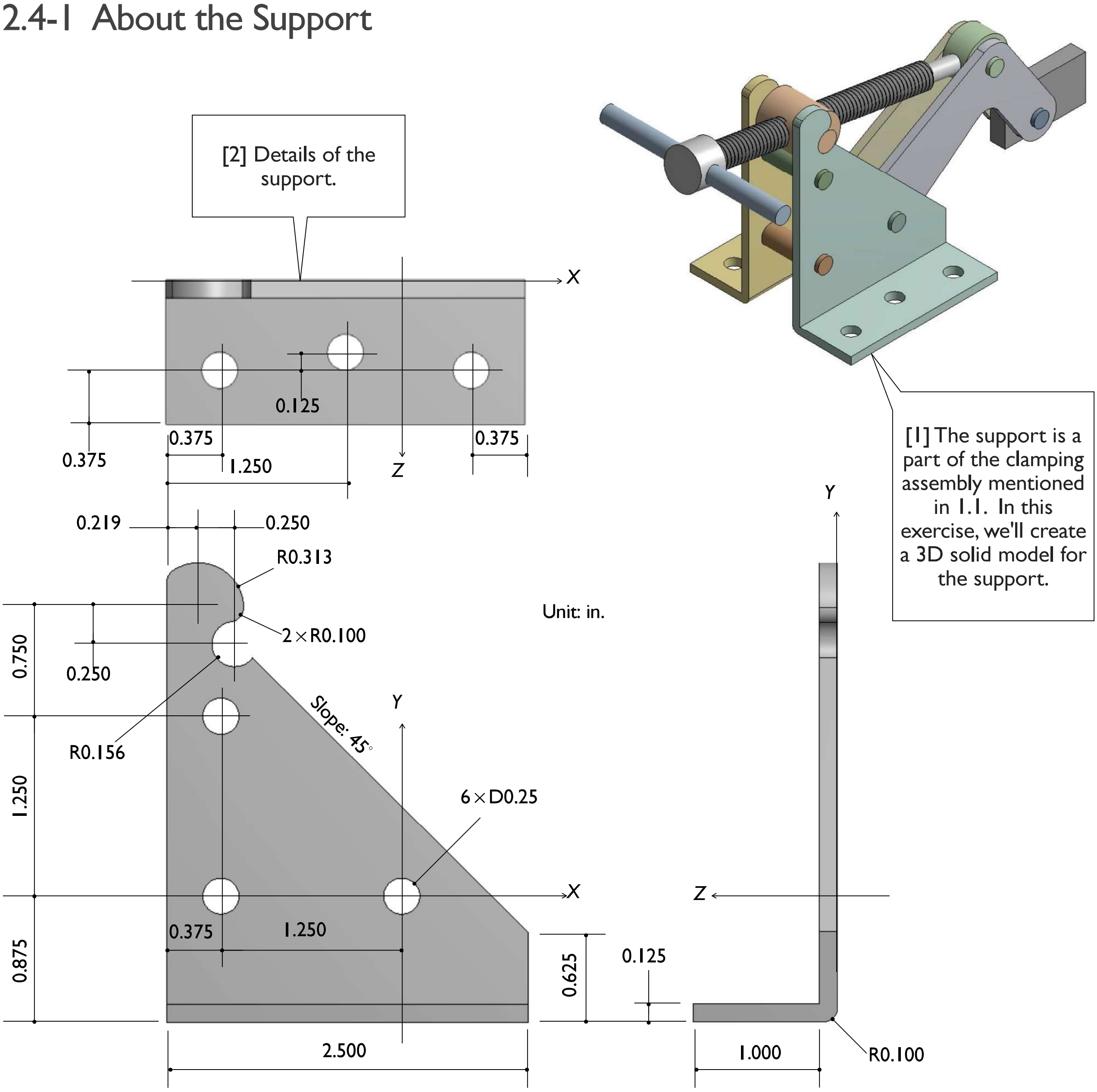
[10] Save the part with the file name **Yoke**. Close the file and exit **SolidWorks**.

Section 2.4

Support



2.4-1 About the Support



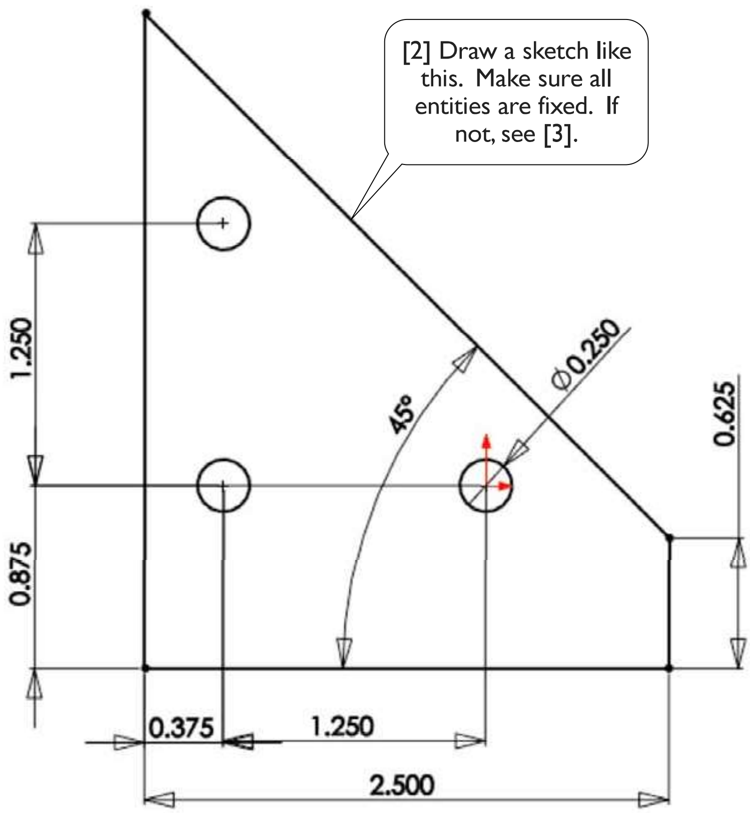
2.4-2 Start Up

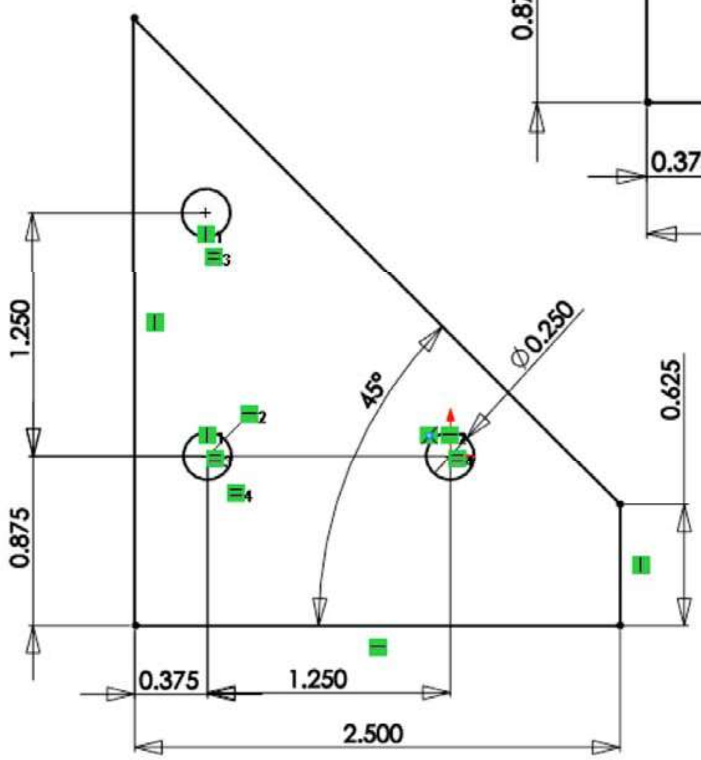
[1] Launch **SolidWorks** and create a new part. Set up **IPS** unit system and with 3 decimal places for the length unit.

2.4-3 Create Vertical Plate

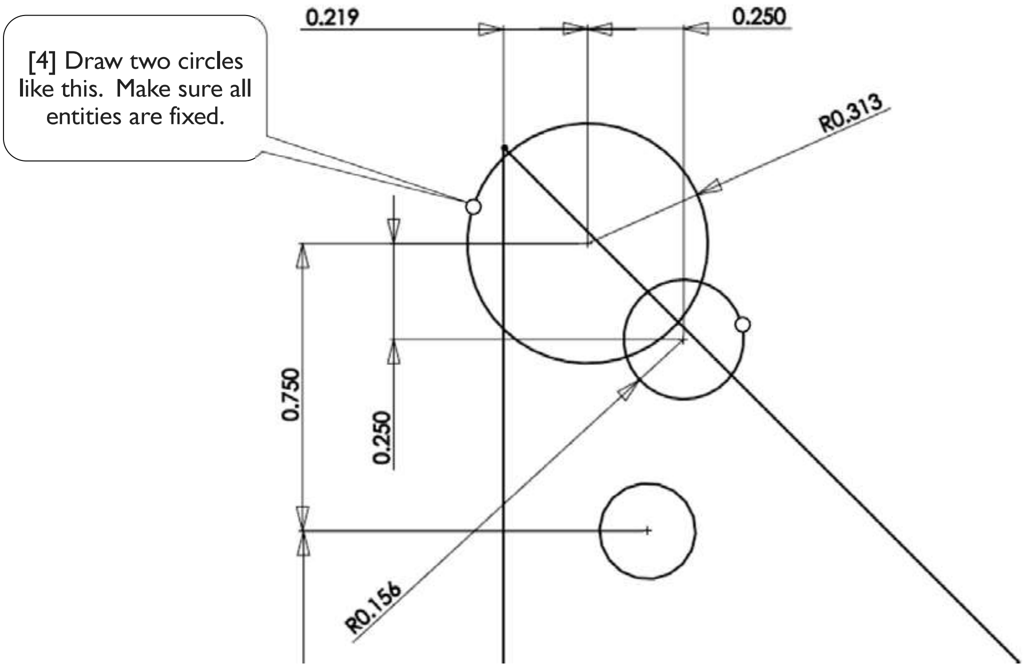
[1] Create a sketch on **Front** plane.

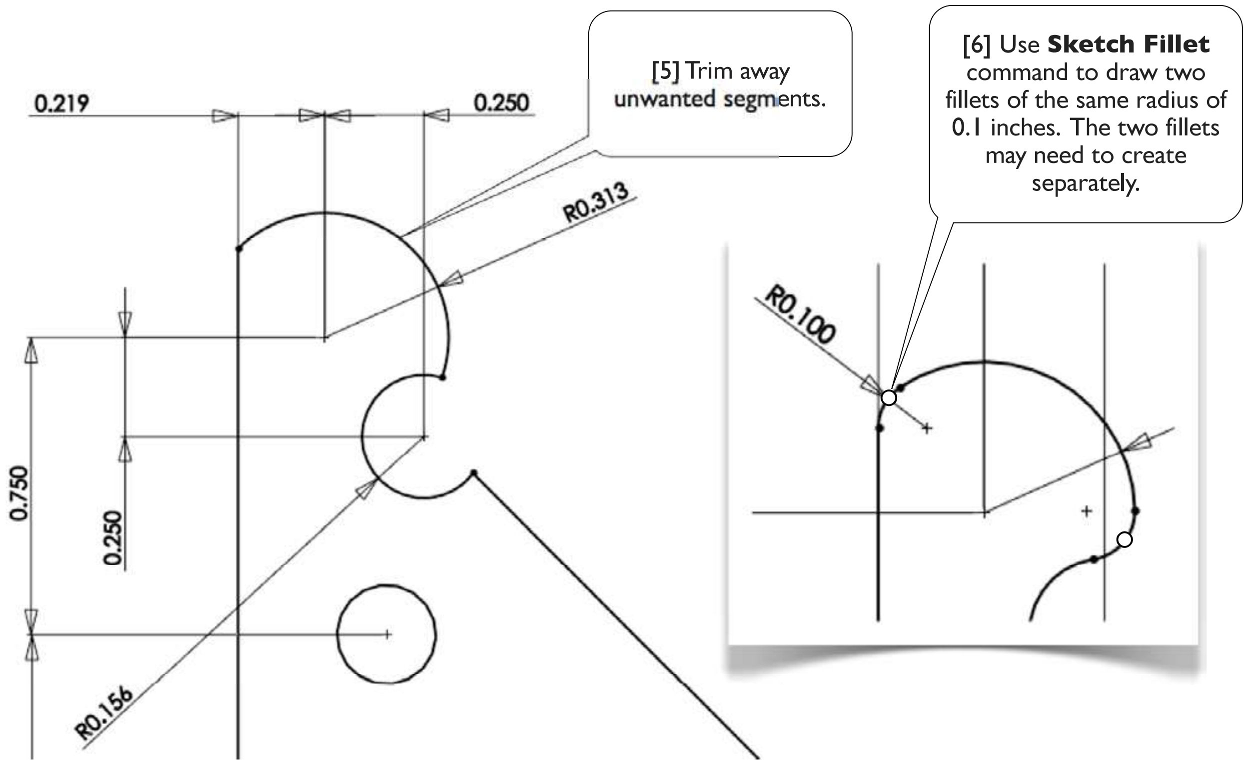
[2] Draw a sketch like this. Make sure all entities are fixed. If not, see [3].

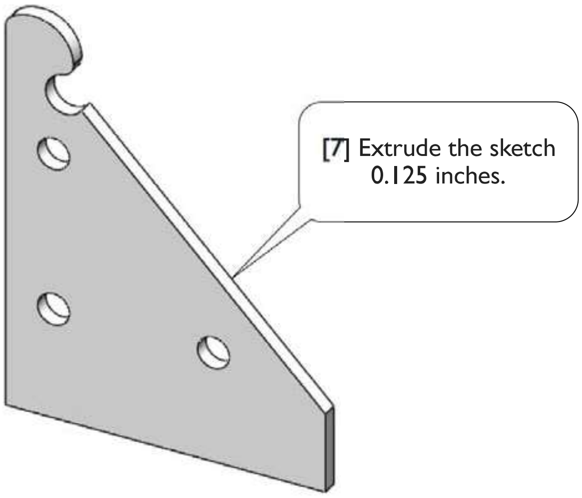




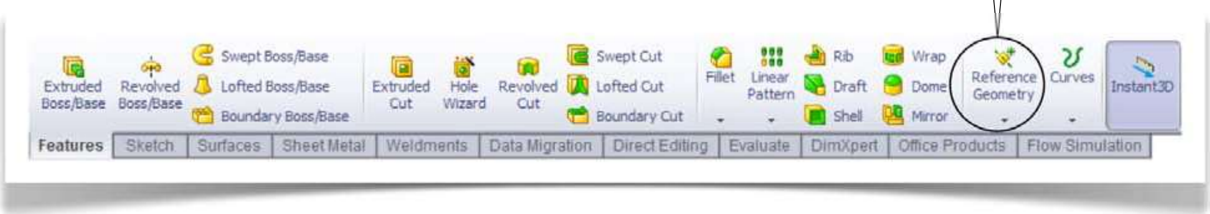
[3] If there are any blue-colored entities, add missing **Relations**. Remember, one way to detect missing **Relations** is to drag an unfixed entity.

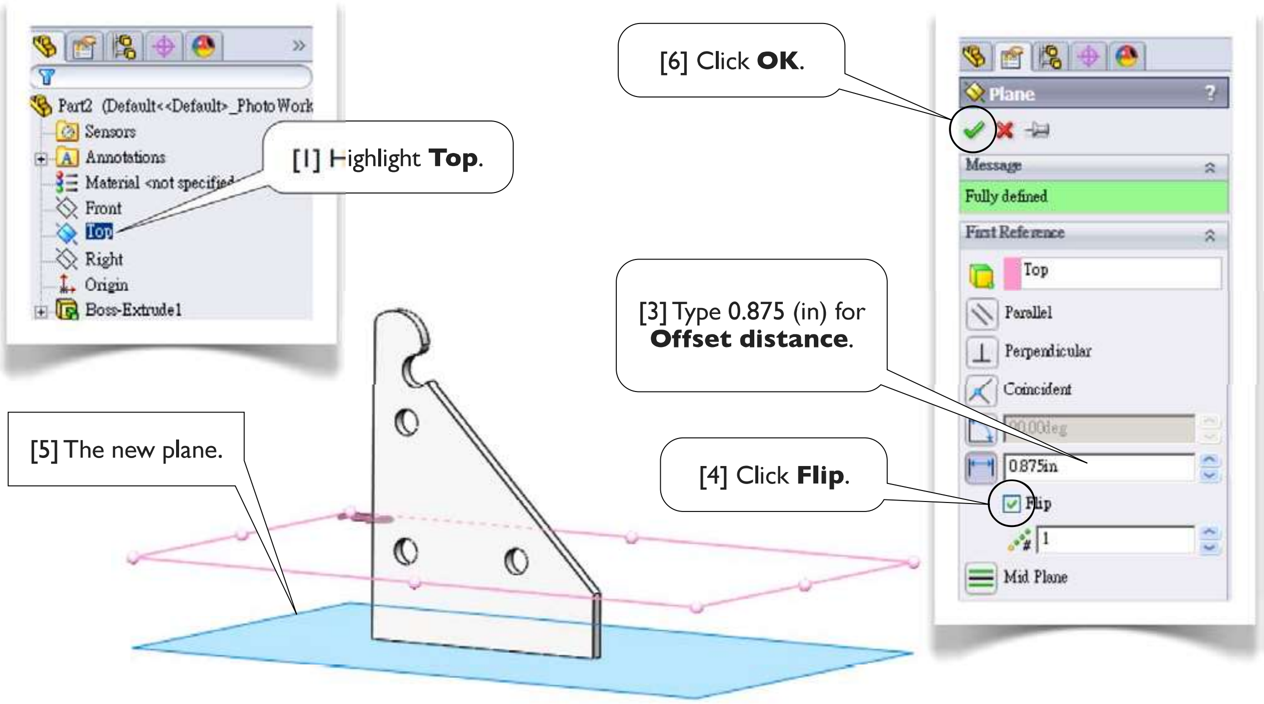

[4] Draw two circles like this. Make sure all entities are fixed.
0.219
0.250
R0.313
0.750
0.250
R0.156

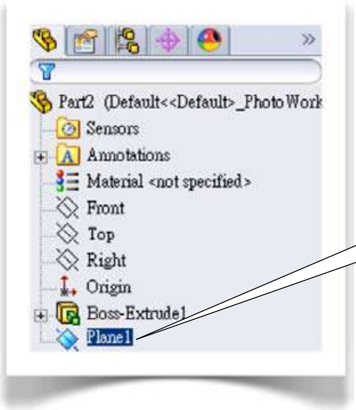

[5] Trim away unwanted segments.
0.219
0.250
R0.313
0.750
0.250
R0.156
[6] Use **Sketch Fillet** command to draw two fillets of the same radius of 0.1 inches. The two fillets may need to create separately.
R0.100



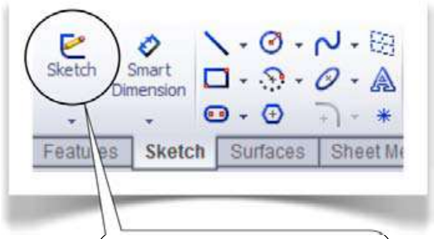
2.4-4 Create Horizontal Plate







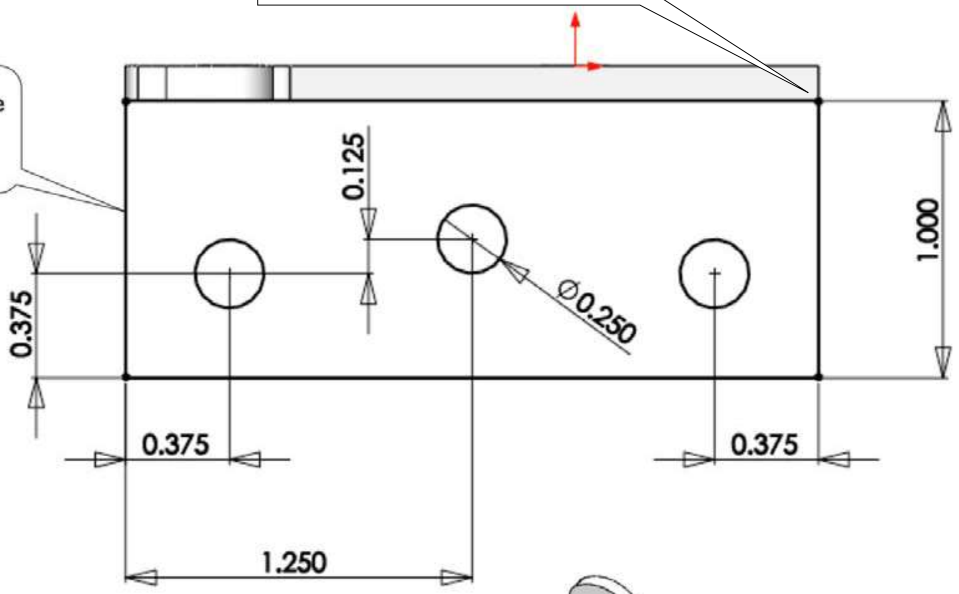
[7] While the new plane is still highlighted, click **Sketch** in the **Sketch Toolbar** (see [8]).

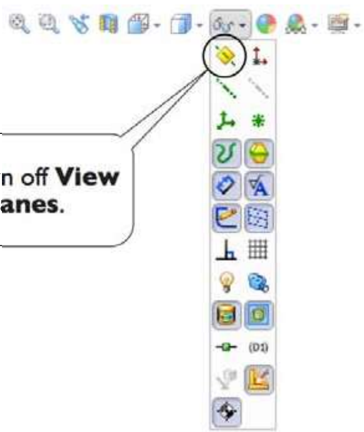


[8] Click **Sketch** in the **Sketch Toolbar**.

[10] Remember, one way to detect missing **Relations** is to drag an unfixed entity. You may need to add a **Coincident** relation here.

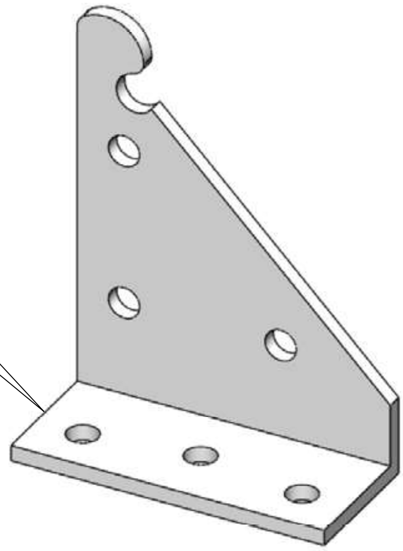
[9] Draw a sketch like this. Make sure all entities are fixed.





[12] Turn off **View Planes**.

[11] Extrude (upward) 0.125 inches.



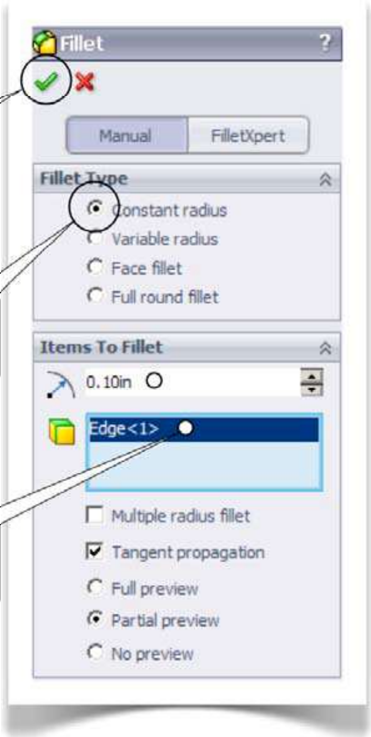
2.4-5 Create Fillet

[1] In the **Features Toolbar**, click **Fillet**.

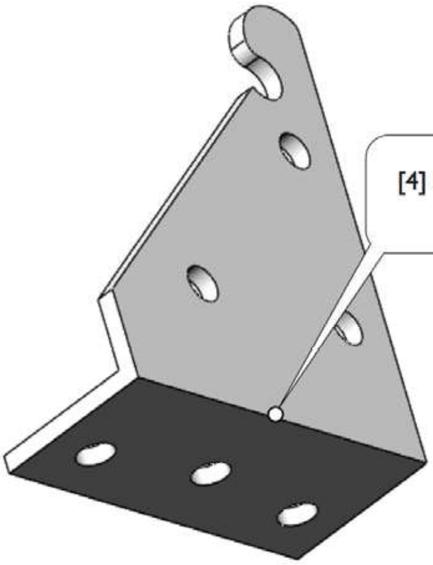
[5] Click **OK**.

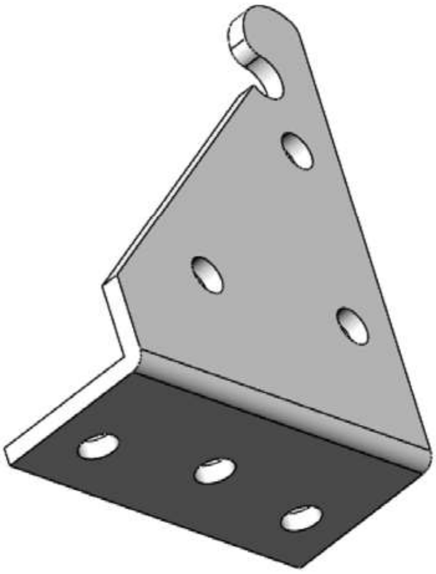
[2] Select **Constant radius**.

[3] Select the edge shown in [4].



[4] Select this edge.





[6] Save the part with the file name **Support**. Close the file and exit **SolidWorks**.