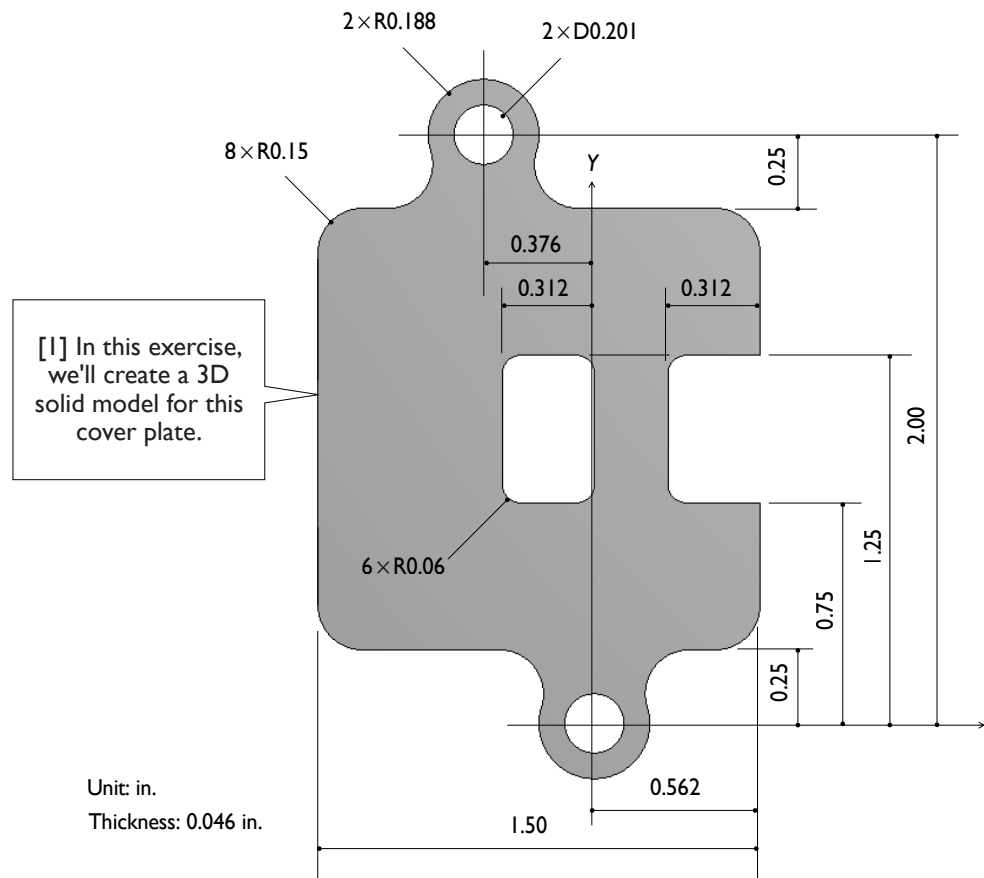


# Section 1.4

## Cover Plate



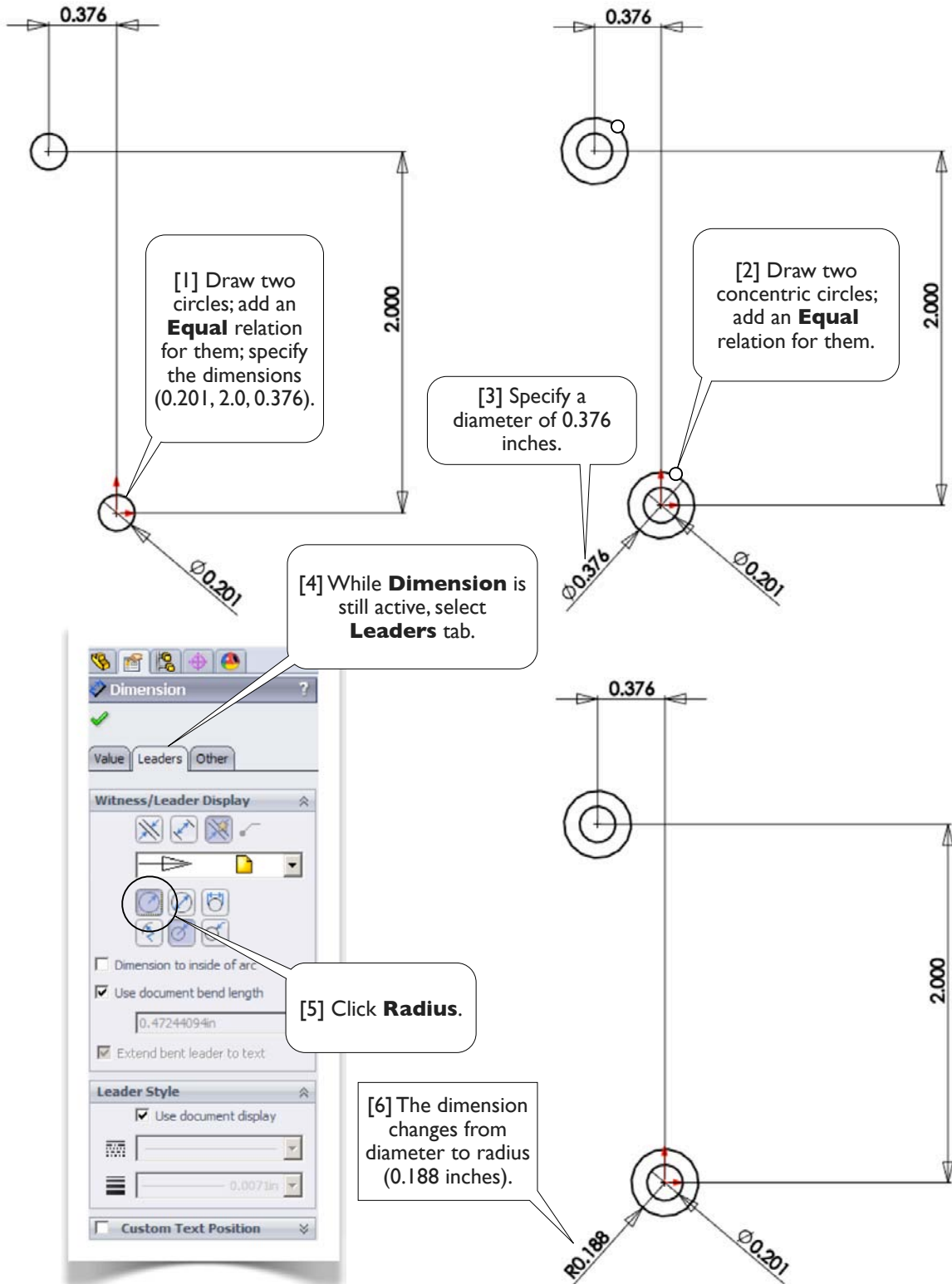
### 1.4-1 About the Cover Plate



### 1.4-2 Start Up

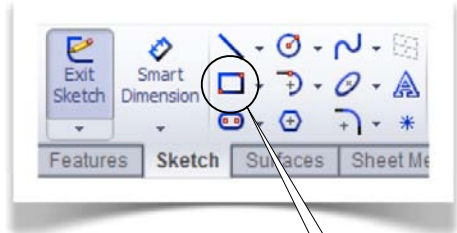
[1] Launch **SolidWorks** and create a new part (1.1-2). Set up **IPS** unit system with 3 decimal places for the length unit (1.1-3). Create a sketch on **Front** plane (1.1-4[1, 2]).

### I.4-3 Draw the Sketch



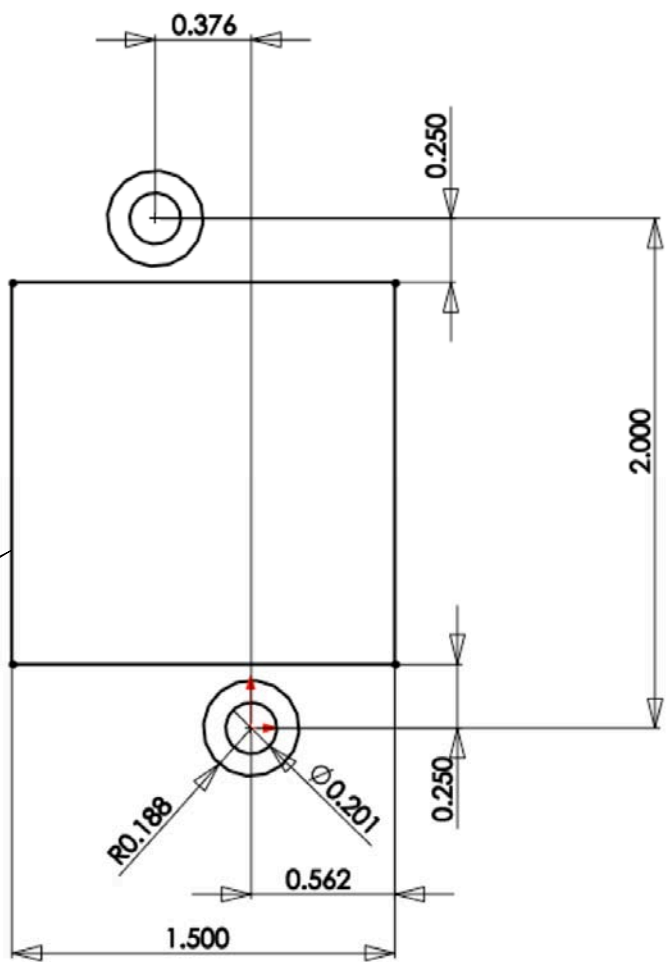


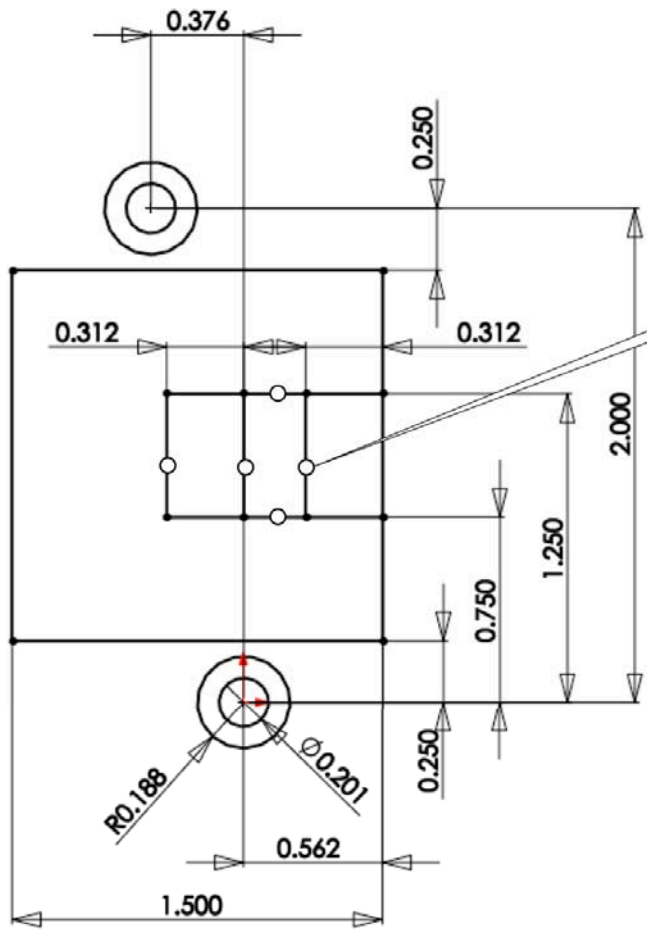
[7] Select **Corner Rectangle** from the **Context Menu**.



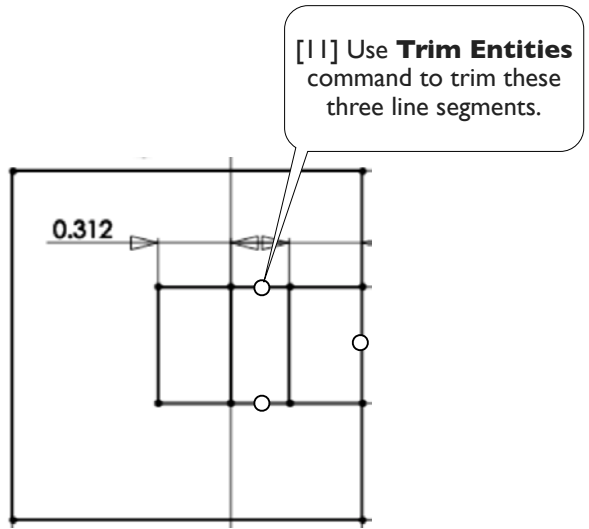
[8] The **Corner Rectangle** is also available in the **Sketch Toolbar**.

[9] Draw a rectangle and specify the dimensions (1.5, 0.25, 0.25, 0.562).

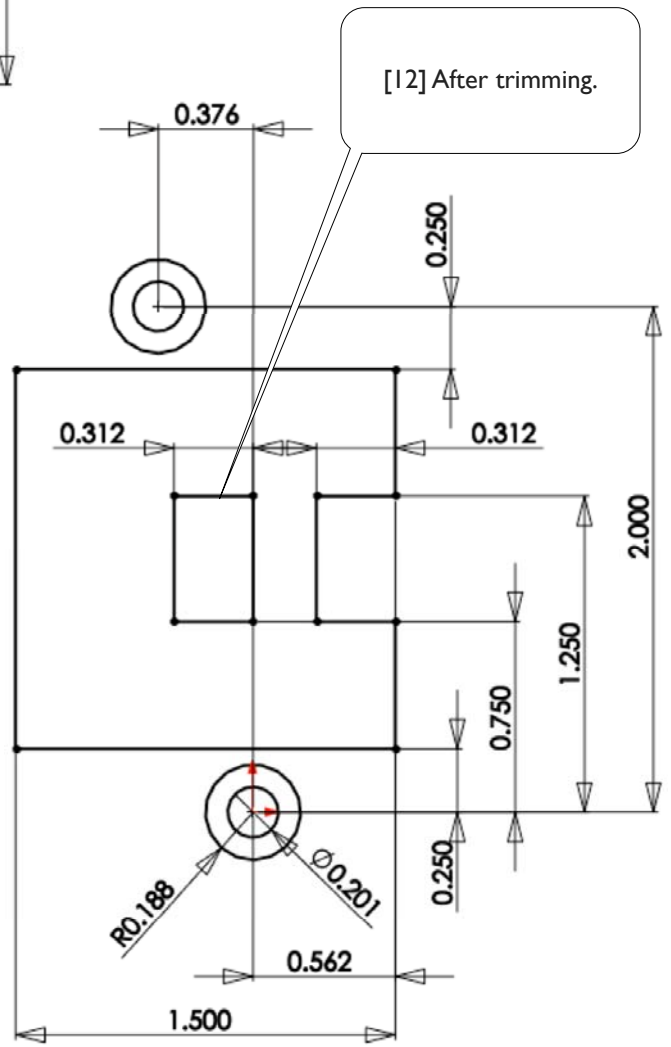




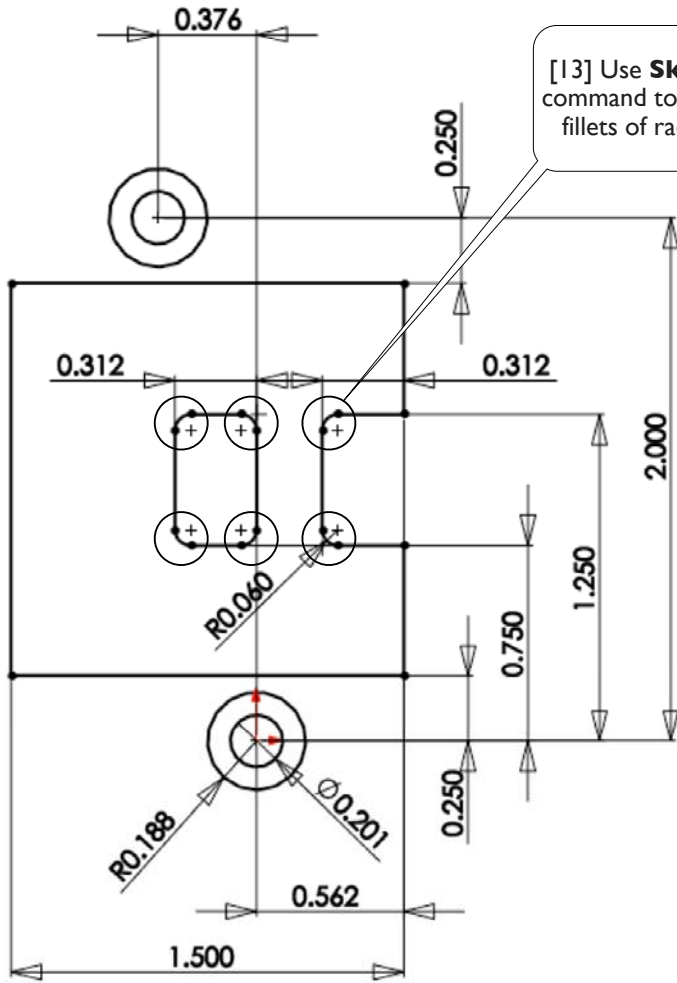
[10] Use **Line** command to draw these 5 line segments; specify the dimensions (0.75, 1.25, 0.312, 0.312).



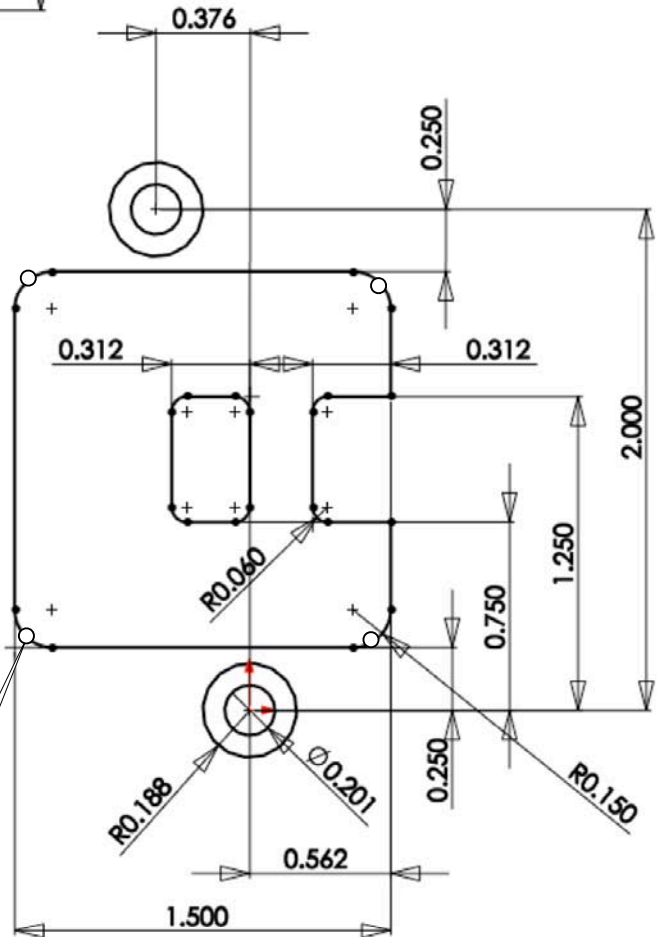
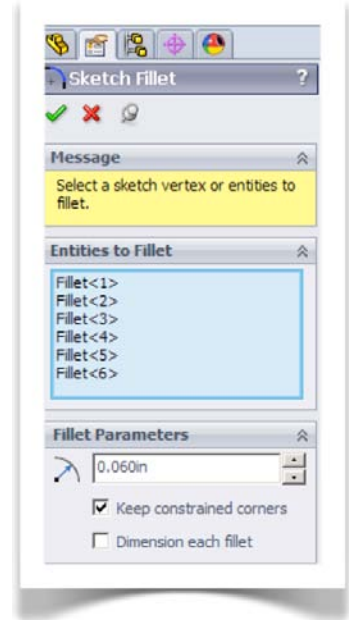
[11] Use **Trim Entities** command to trim these three line segments.



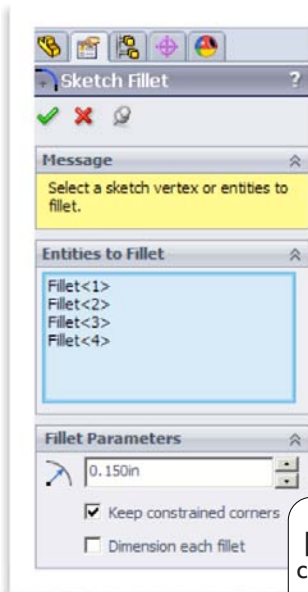
[12] After trimming.



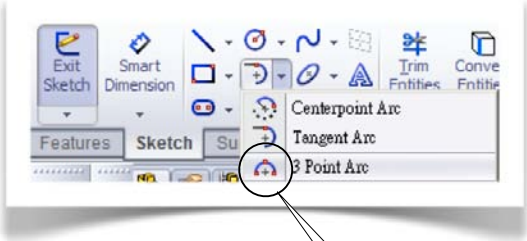
[13] Use **Sketch Fillet** command to draw these 6 fillets of radius 0.06 in.



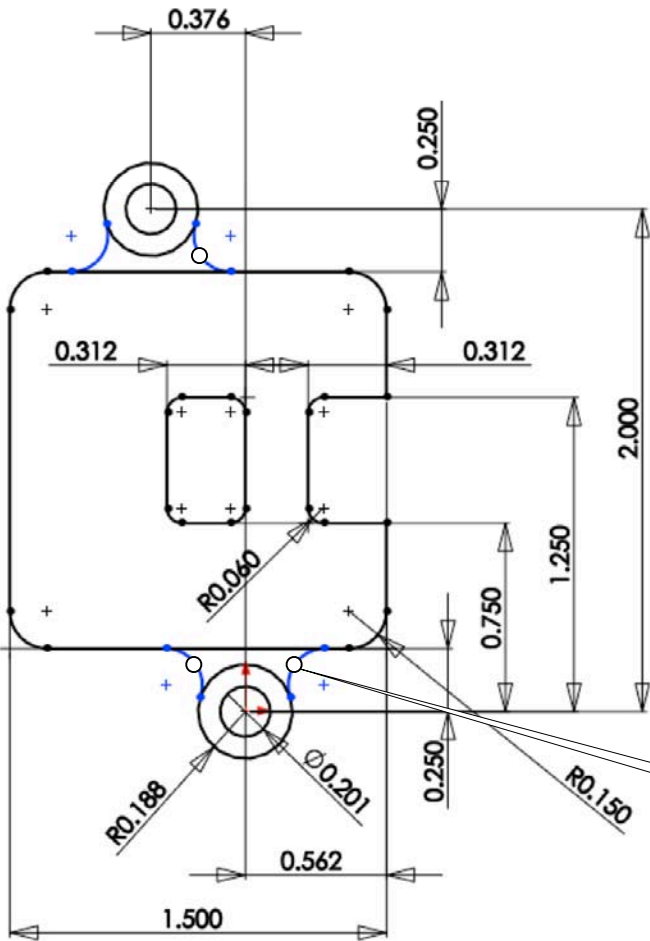
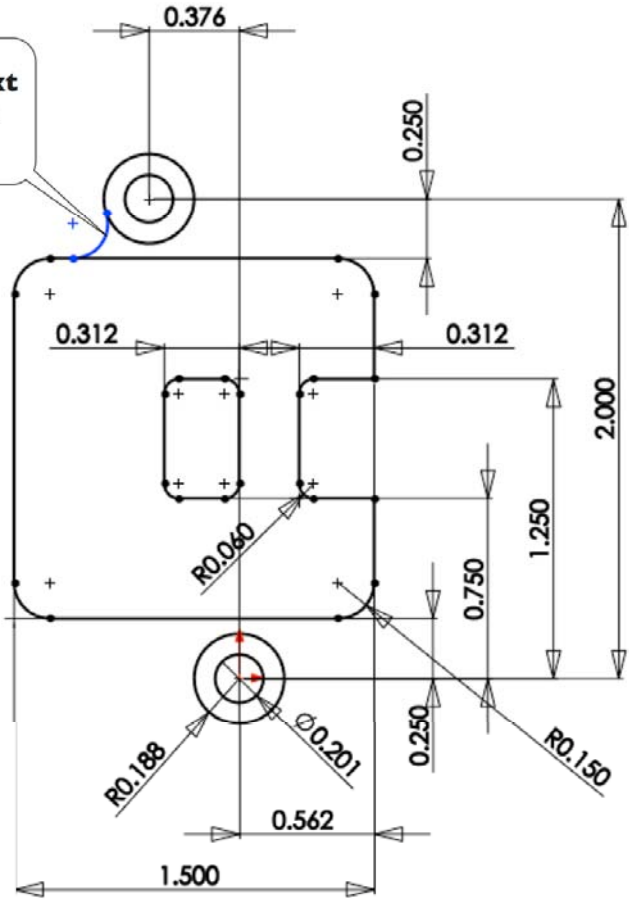
[14] Use **Sketch Fillet** command to draw these 4 fillets of radius 0.15 in.



[15] Select **3 Point Arc** from the **Context Menu** and draw an arc like this. Add two **Tangent** relations.

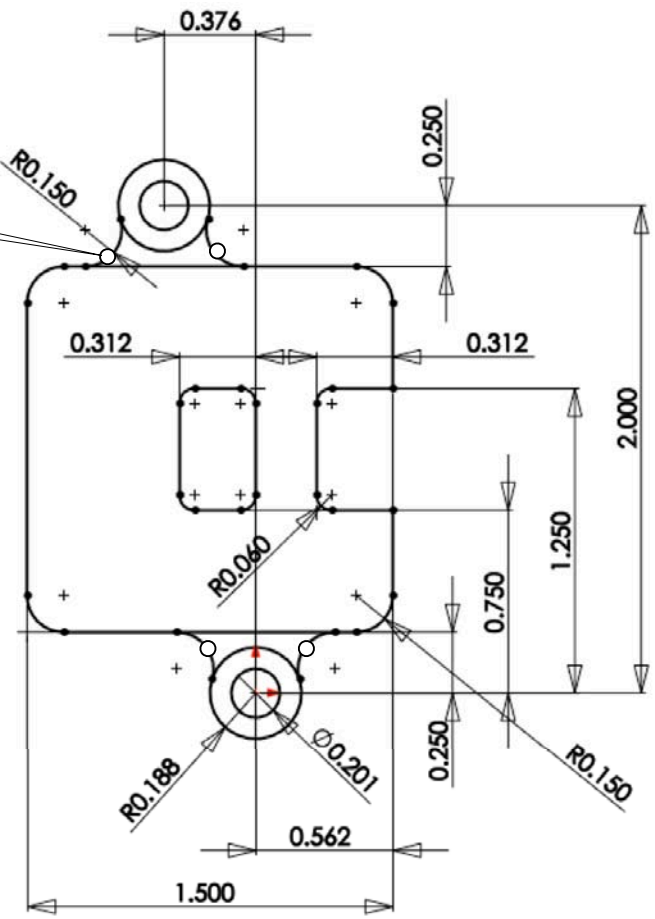


[16] The **3 Point Arc** command is also available in the **Sketch Toolbar**.

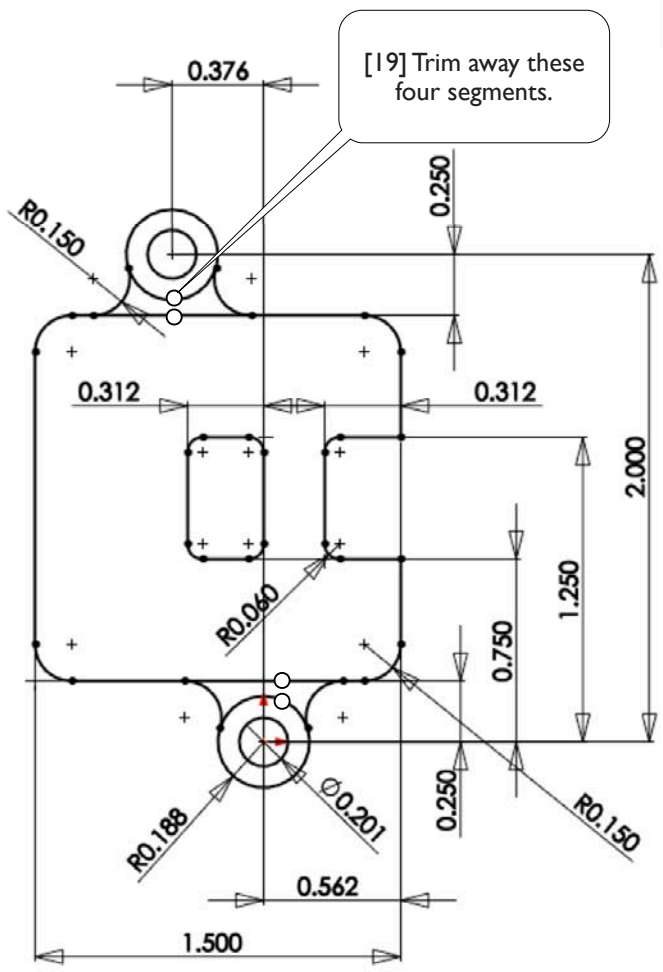


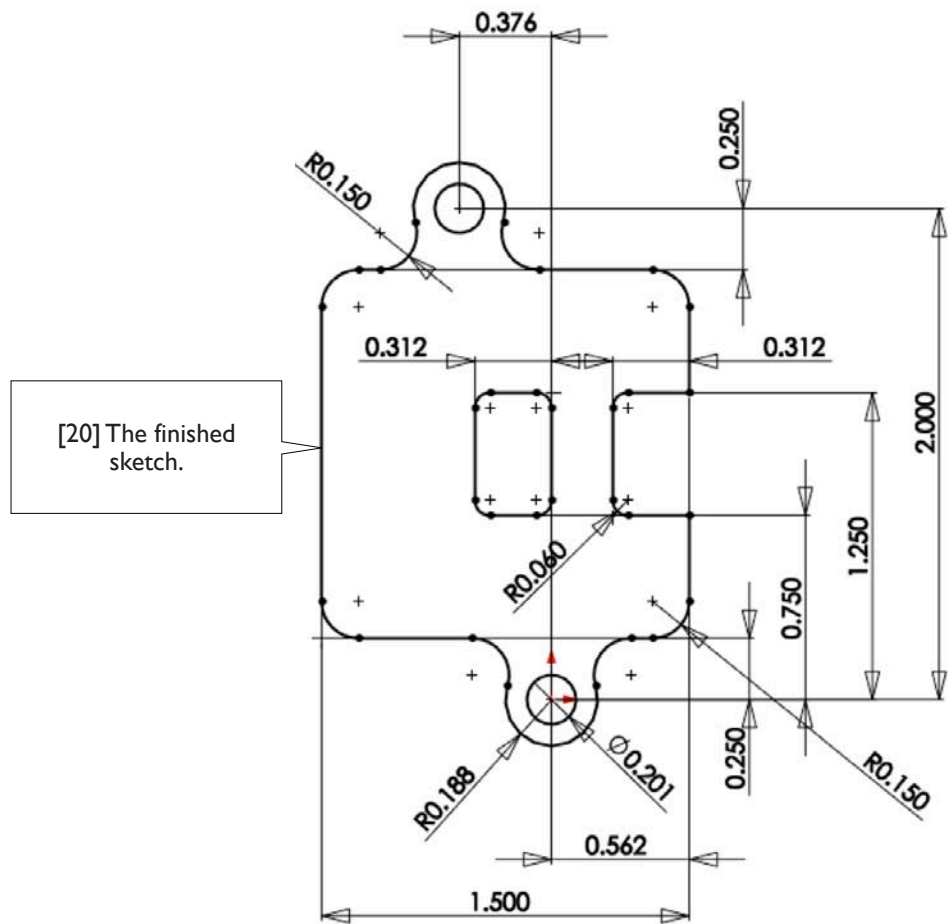
[17] Repeat step [15] three more times.

[18] Add an **Equal** relation for these four arcs. Specify a radius dimension of 0.15 inches.



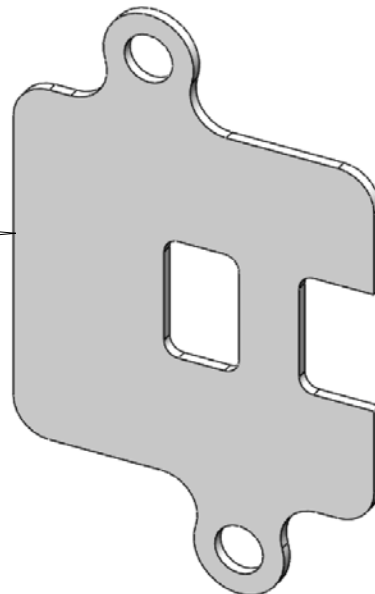
[19] Trim away these four segments.





## I.4-4 Generate 3D Model

[1] **Extrude** the sketch 0.046 inches to create this 3D model.



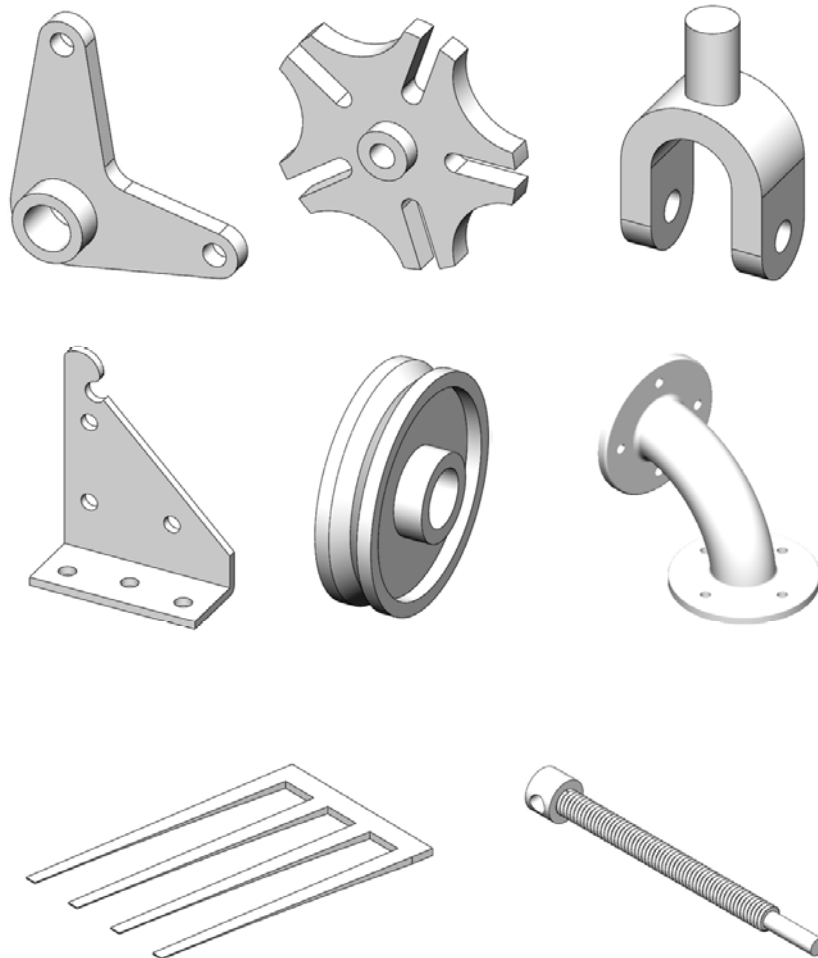
[2] Save the part with the file name **Cover**. Close the file and exit **SolidWorks**.



# Chapter 2

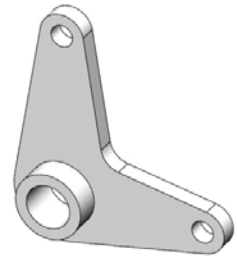
## Part Modeling

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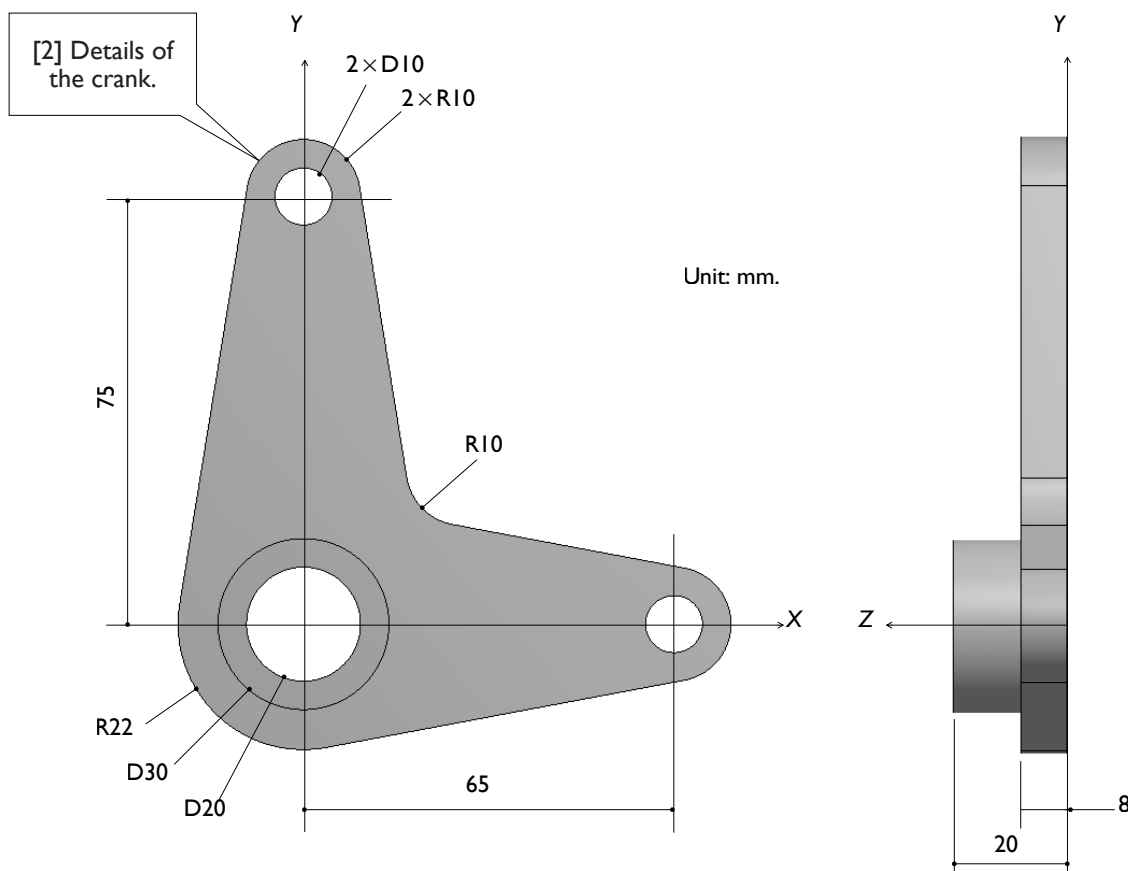
# Section 2.1

## Crank



### 2.1-1 About the Crank

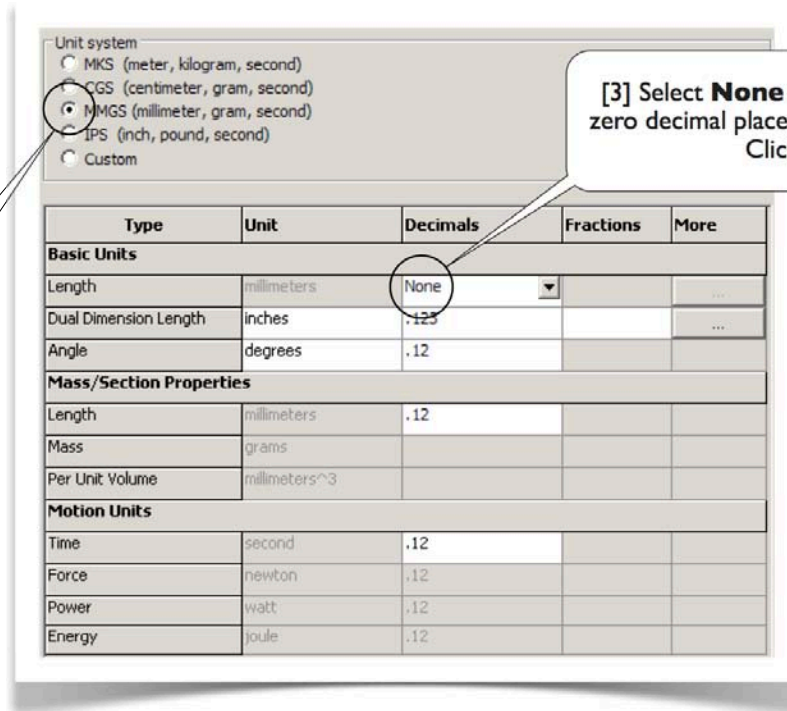
[1] In this exercise, we'll create a 3D solid model for a crank [2]. The crank model can be viewed as a series of three two-step operations; each involves drawing a sketch on a plane and then extruding the sketch to generate a solid body. The material of the body is either added to the existing body or cut from the existing body.



## 2.1-2 Start Up

[1] Launch **SolidWorks** and create a new part.

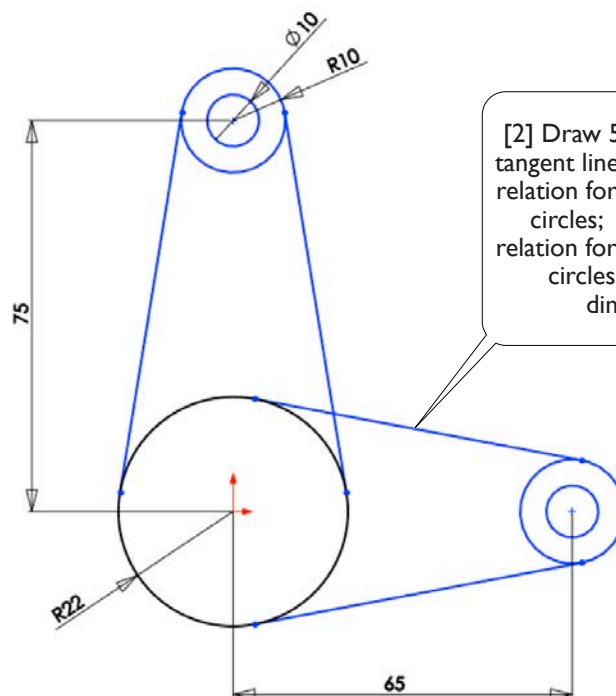
[2] In the **Options**, select **MMGS** as unit system.



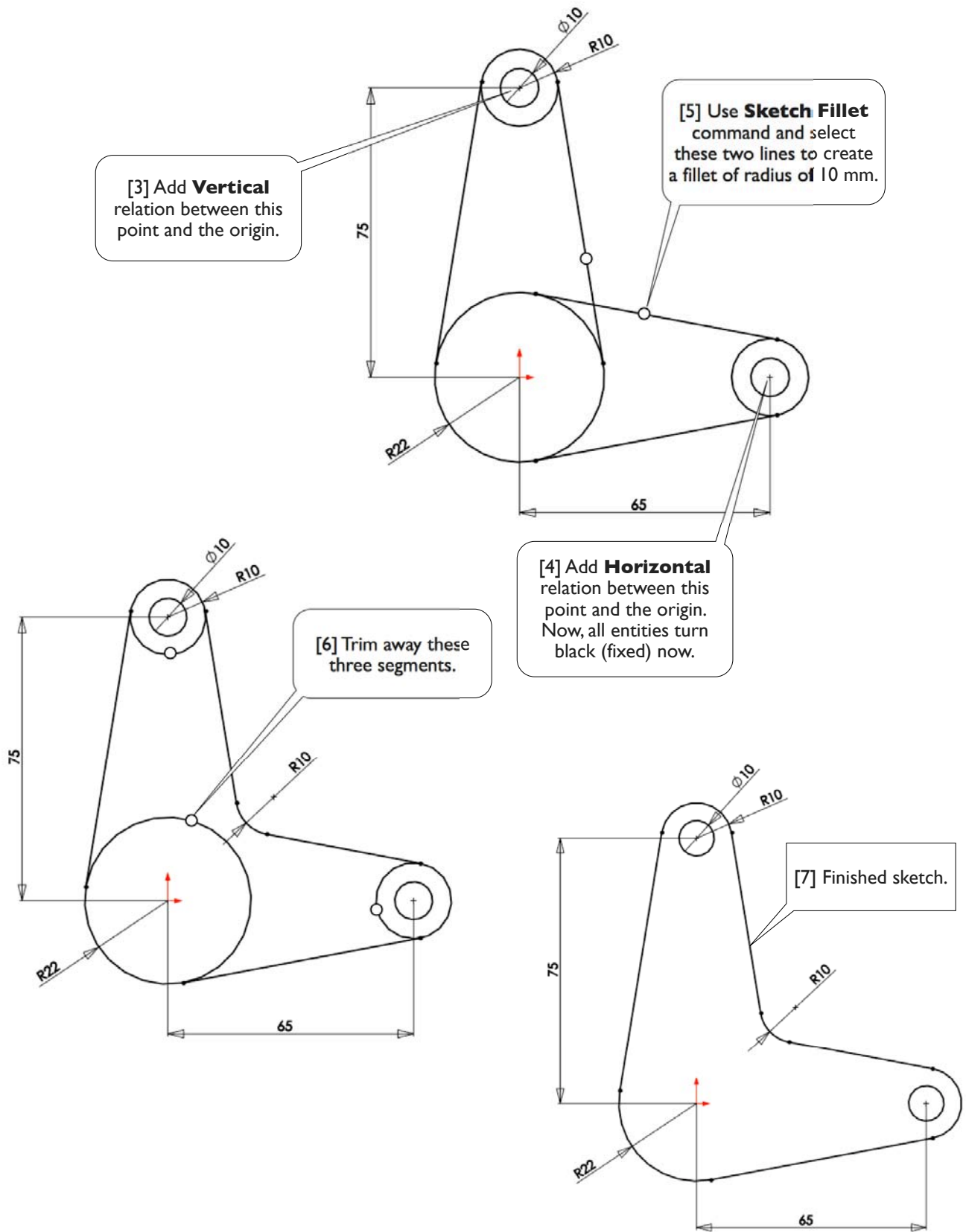
[3] Select **None** for **Decimals** (i.e., zero decimal places for the length unit). Click **OK**.

## 2.1-3 Draw a Sketch for the Base Body

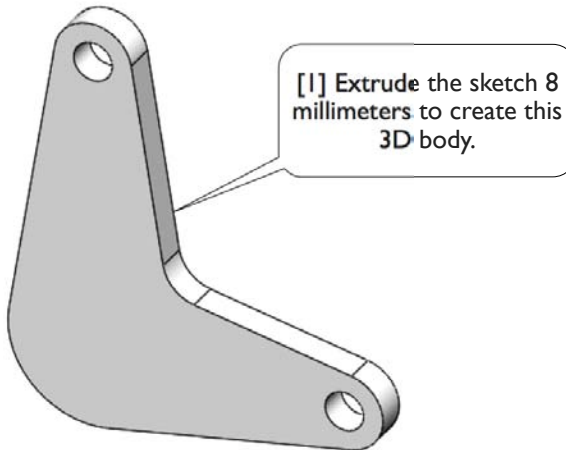
[1] Create a sketch on **Front** plane.



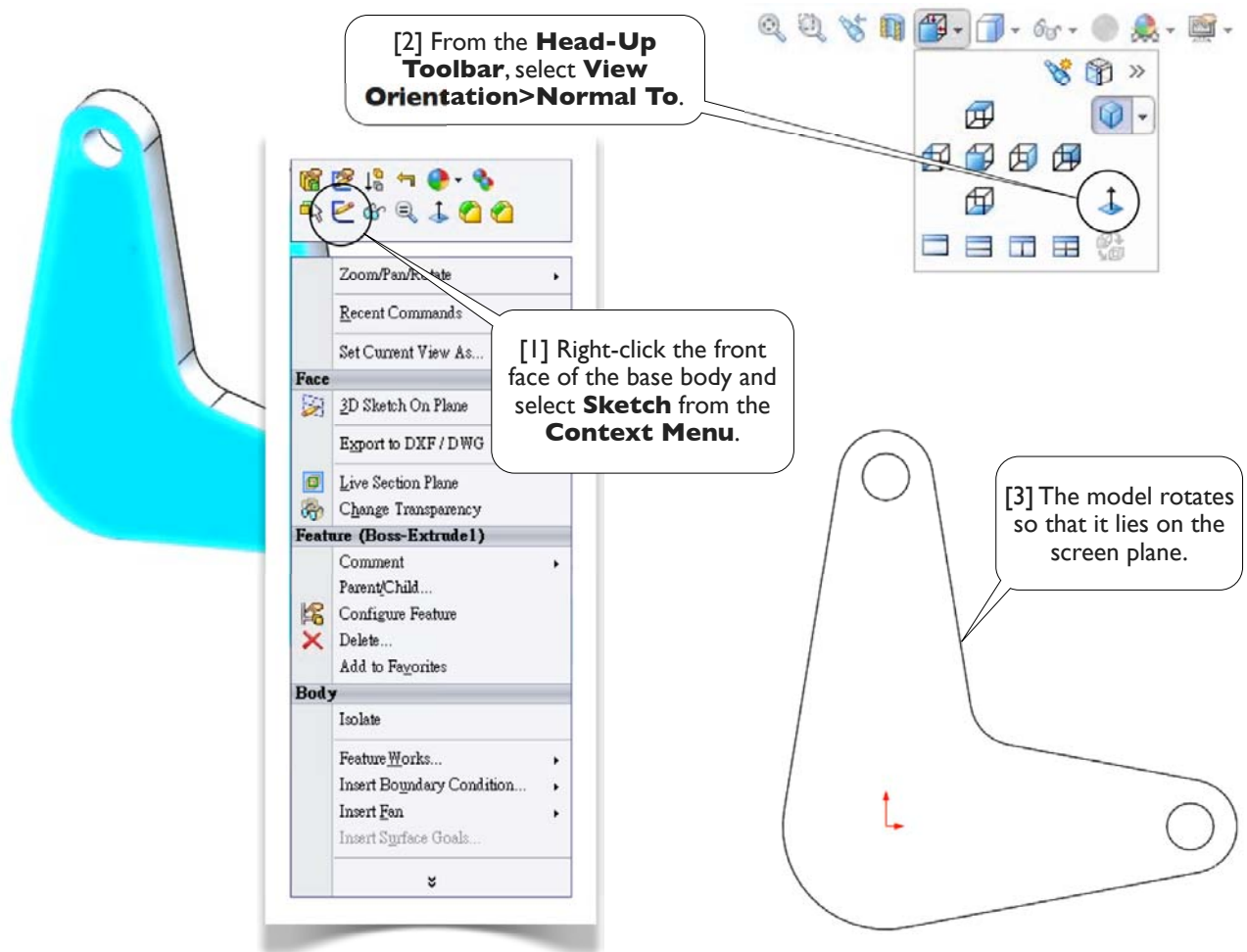
[2] Draw 5 circles and four tangent lines. Add an **Equal** relation for the two smallest circles; add an **Equal** relation for the two medium circles. Specify the dimensions.



## 2.1-4 Extrude the Sketch to Create the Base Body



## 2.1-5 Add Features to the Base Body

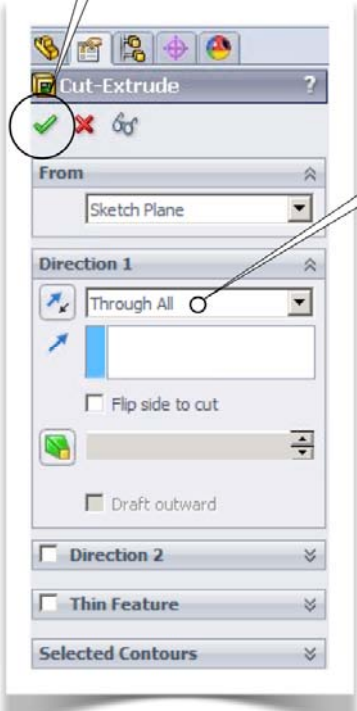






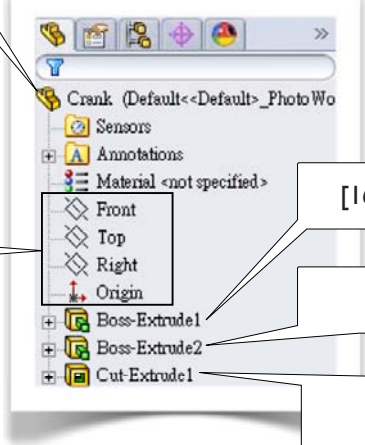
[9] From the **Pull-Down Menus**, select **Insert>Cut>Extrude...** or, from **Features Toolbar**, select **Extruded Cut**.

[11] Click **OK**.



[10] Select **Through All** for **End Condition**.

[14] The **Features Tree**.

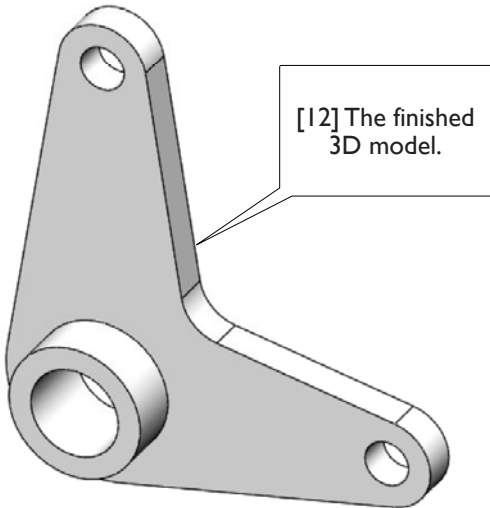


[15] The **reference geometries**.

[16] The base body.

[17] The boss.

[18] The hole.



[12] The finished 3D model.

[19] A **part** consists of reference geometries, base body, and **features** on the base body. In this example, the features added to the base body are the boss and the hole. Note that it is equally good to treat the two small holes as features on the base body.

[13] Save the part with the file name **Crank**.

[20] Exit **SolidWorks**.