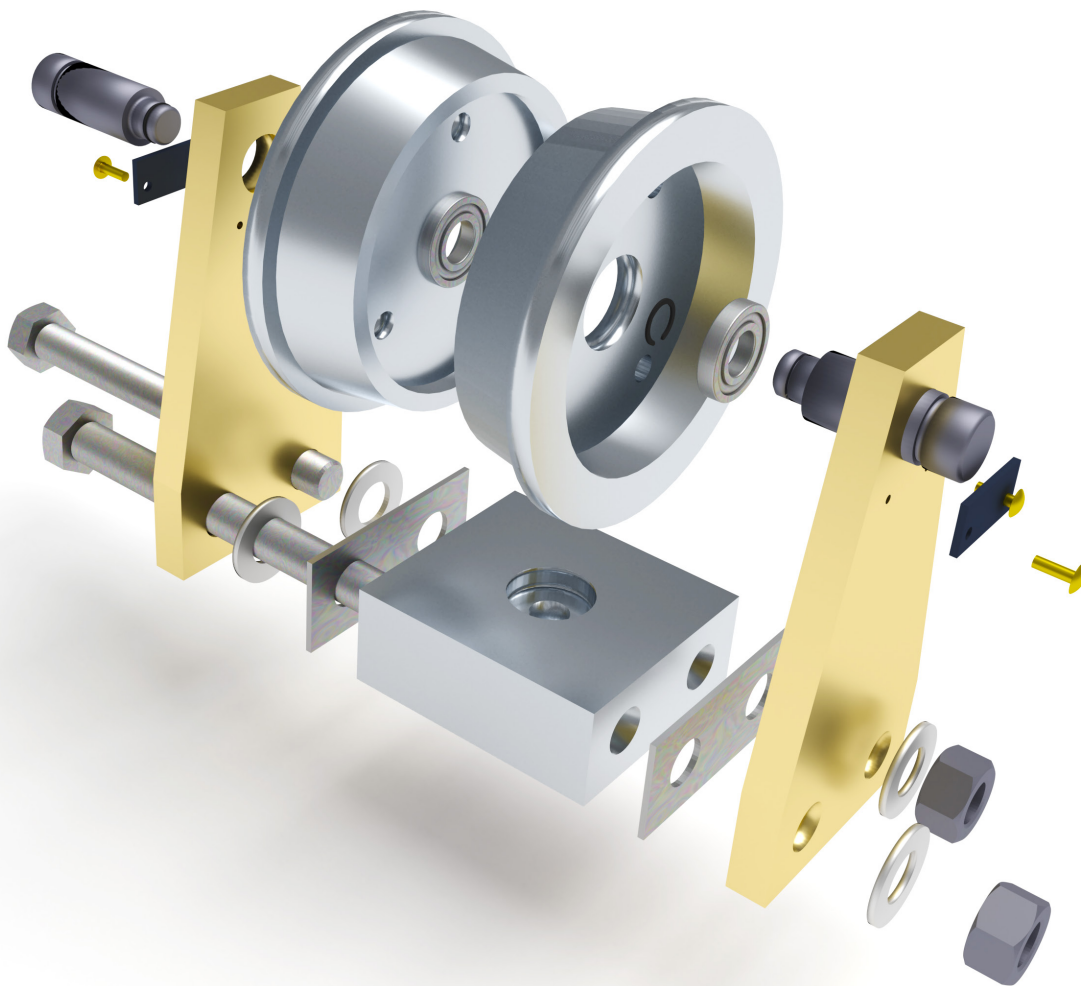


# Project Based SOLIDWORKS® 2019

**Videos**  
Includes over nine  
hours of video instruction



Kirstie Plantenberg, CSWA

Visit the following websites to learn more about this book:



**amazon.com**

**Google books**

**BARNES & NOBLE**

## CHAPTER 2

---

# CONNECTING ROD PROJECT Model

### CHAPTER OUTLINE

2.1) PREREQUISITES .....	2
2.2) WHAT YOU WILL LEARN .....	2
2.3) SETTING UP THE PROJECT .....	3
2.4) BASE EXTRUDE.....	3
2.5) ADDING FEATURES .....	10
2.6) ADDING MATERIAL .....	14
CONNECTING ROD PROJECT (MODEL) PROBLEMS .....	17
CONNECTING ROD PROJECT (MODEL) QUIZ PROBLEMS .....	27

## 2.1) PREREQUISITES

Before completing this tutorial, you should have completed the following tutorial and be familiar with the following topics.

### Pre-requisite Tutorial

- Chapter 1 – Introduction to SOLIDWORKS®

### Pre-requisite Topics

- Computer navigation.
- Passing familiarity with orthographic projection.
- Ability to read dimensions.

## 2.2) WHAT YOU WILL LEARN

The objective of this tutorial is to introduce you to creating simple *Sketches*, *Extrudes* and *Cuts*. You will be modeling the connecting rod shown in Figure 2.2-1. Specifically, you will learn the following commands and concepts.

### Sketching

- Sketch relations
- Editing dimensions
- Editing sketches
- Sketch chamfers
- Sketch fillet
- Rectangle

### Features

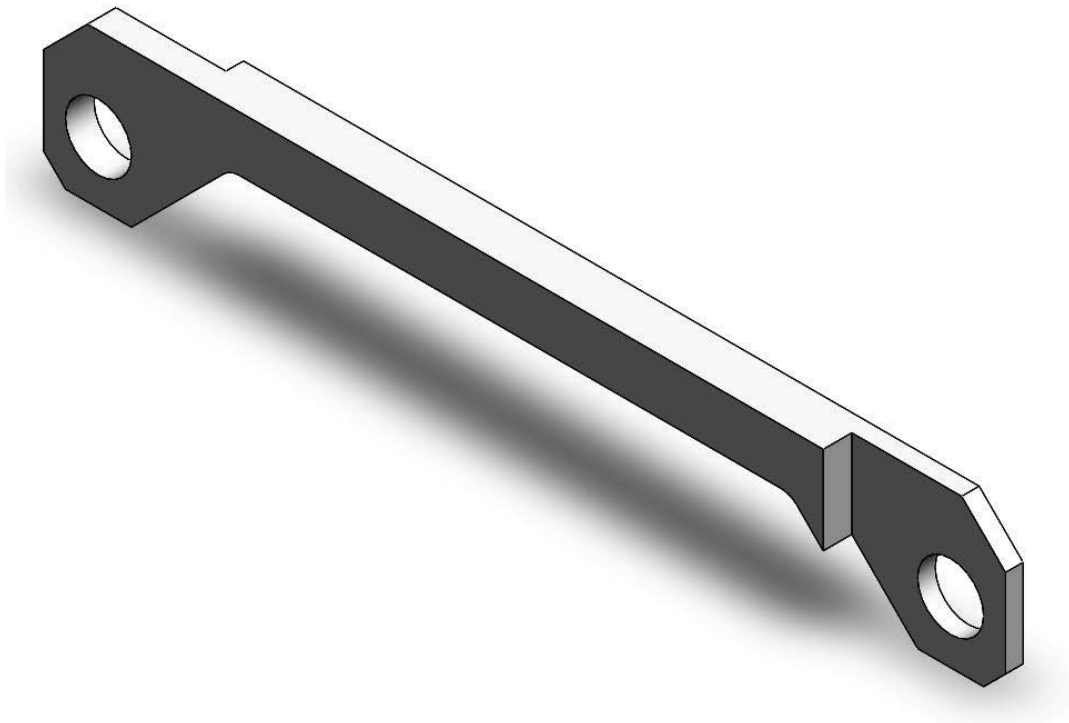
- Chamfer
- Fillets
- Editing a feature

### Material and properties

- Applying material
- Mass properties

### View

- Panning
- Rotating


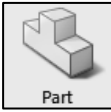



---

Figure 2.2-1: Connecting rod


---

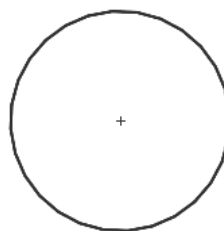
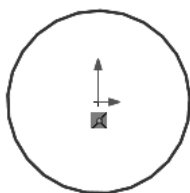
## 2.3) SETTING UP THE PROJECT

- 1) **Start SOLIDWORKS** and then start a new  **part** .
- 2) Set your unit to **IPS** (i.e. inch, pound, second) and set your **Decimals = .12** and your standard to **ANSI** (*Options*  – *Document Properties – Units*).
- 3) Save your part as **CONNECTING ROD.SLDPR**T (**File – Save**). Remember to save often throughout this project.

## 2.4) BASE EXTRUDE

- 1) **Sketch**  on the **Front Plane**.

- 2) Use the **Circle**  **Circle** command to sketch two circles as shown below. Make one of the circle centers **coincident** with the origin. You will know when you have snapped to the origin when a small circle appears. Don't worry about the circle's spacing or size at the moment. Just make them about 5 or more diameters apart.



- 3) **Pan** your drawing area to center the circles. (See the informational block on **Panning**.)


### Panning

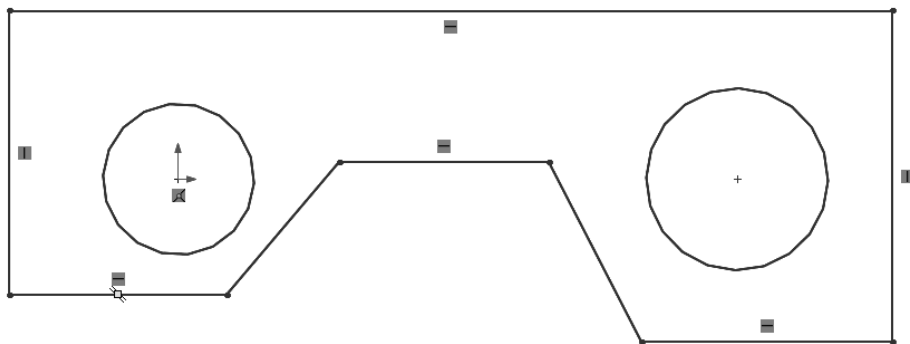
#### Method 1

- 1) Hold down the **Ctrl** key.
- 2) Click and hold your **middle mouse button**.
- 3) **Move** your mouse.

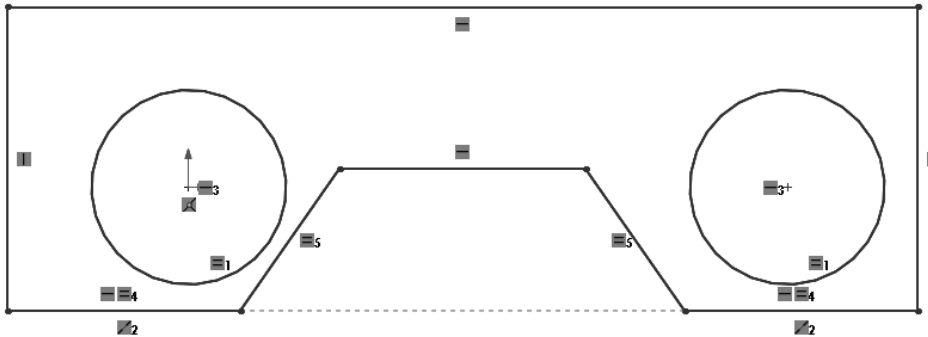
#### Method 2

- 1) Select **View – Modify - Pan**  commands in the *View* toolbar.
- 2) Click your **right mouse button** and move the mouse.

- 4) Use the **Line**  **Line** command to sketch the following profile. Be approximate. Don't worry about getting it exact. Notice that when you are drawing the lines that dashed lines will appear occasionally. These dashed lines allow you to snap to geometric features of the object that have already been drawn or to the origin.



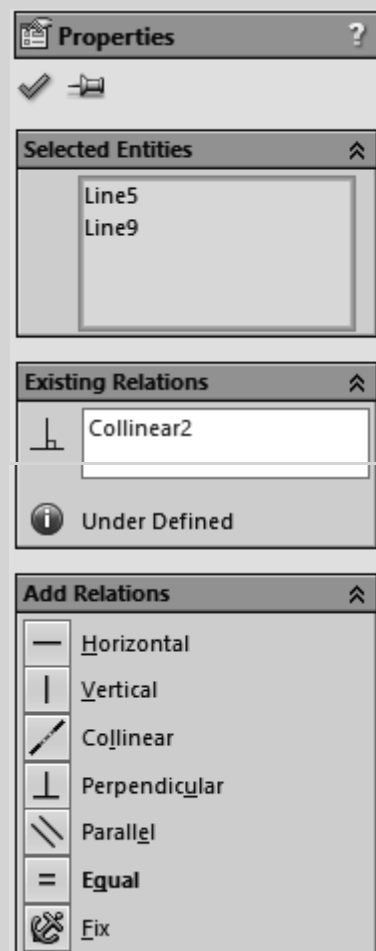
- 5) Add the following sketch relations. Don't worry if your drawing goes wonky. Just click and drag the elements into position. (See the informational block on **Applying Sketch Relations**.)
- If any of the **horizontal** or **vertical** lines are not perfectly horizontal or vertical, add those relations.
  - Make the two circle diameters **Equal**.
  - Make the circle centers **Horizontal**.
  - Make the two bottom horizontal lines **Collinear**.
  - Make the two bottom horizontal line lengths **Equal**.
  - Make the two angled line lengths **Equal**.



### Applying sketch relations

Sketch relations add geometric constraints between two or more entities. For example, we can make two lines parallel, or two circles concentric.

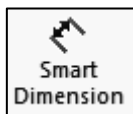
- 1) Select one of the elements that you want to apply the relation to.
- 2) Hold the **Ctrl** key and then select the next element that you want to apply the relation to.
- 3) Continue selecting elements if you want to apply the relation to more than two elements.
- 4) In the *Properties* window, select the relation that you wish to apply.



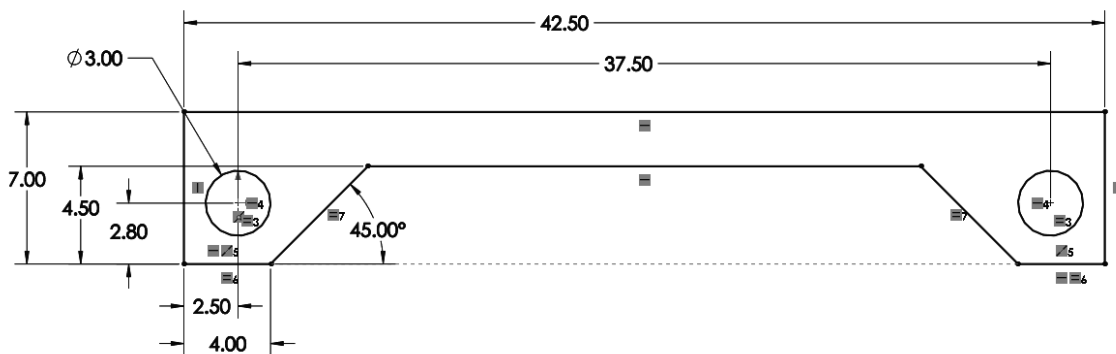
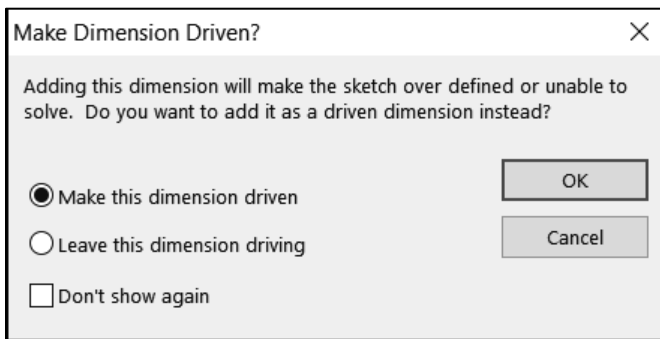
- 6) **View** your **Sketch Relations**. Your geometric relations will show up as symbols inside a green box. (See the informational block on **Viewing Relations**.)

### Viewing relations

- 1) From the pull-down menu at the top, select **View – Hide/Show - Sketch Relations**.




- 7) Add the **Dimensions** shown in the figure shown below. If your drawing exceeds your viewing area, select the **F** key to fit all. When dimensioning, it is a good idea to start with the overall dimensions and then work down to the smaller dimensions. Note that if a *Make Dimension Driven?* window appears, you have an unwanted sketch relation or you have a duplicate dimension. Select **Cancel** and then search and delete the extra constraint. Note that there should be **no blue lines** when you are finished. Blue lines mean that it is under-constrained. Note: If, while you are dimensioning the part, the angled lines become parallel, you need to remove the equal constraint and then reapply it after you have adjusted the lines. (See the informational block on **Editing Dimensions**.)

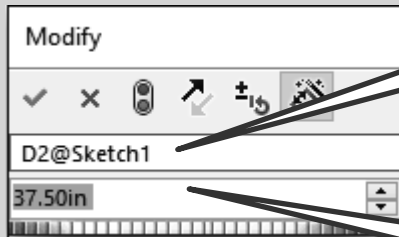




## Editing a dimension

### Method 1

- 1) Double click the dimension that you wish to edit.
- 2) Within the *Modify* window, fill in the correct dimension value and then select the green check mark .

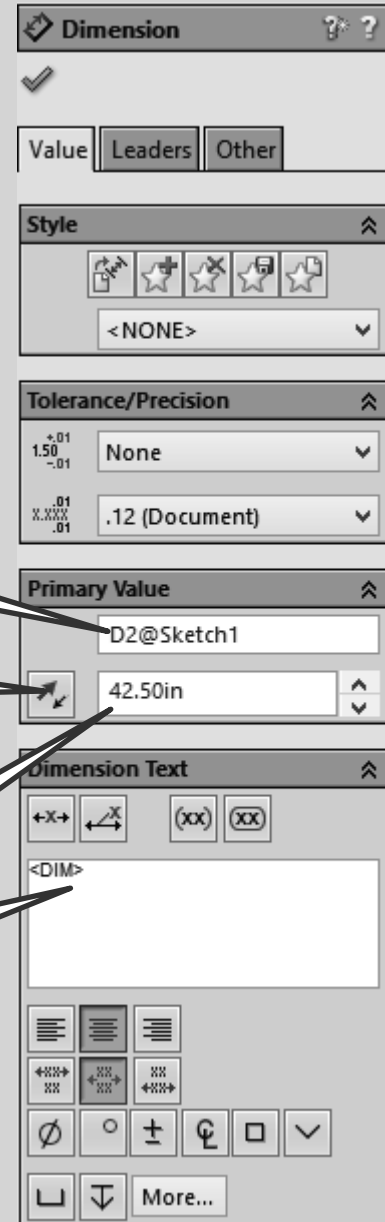


Dimension name

Value

### Method 2

- 1) Select the dimension that you wish to edit.
- 2) Features of the dimension may be changed within the *Dimension* window that appears on the left.

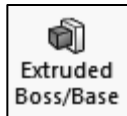


Dimension name

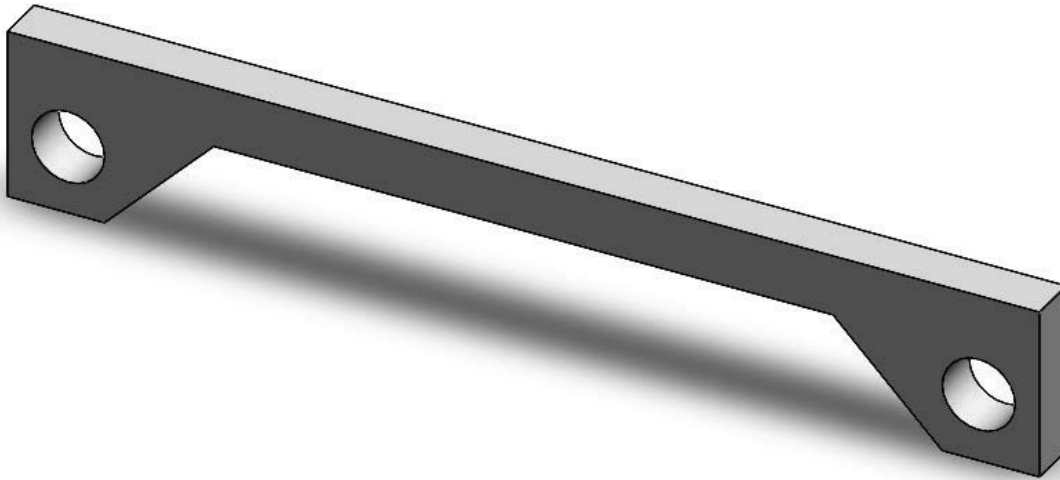
Reverse direction

Value

Add symbols and text



- 8) **Extrude** your sketch to a distance of **2.50** inches.




- 9) Try **zooming in and out** by scrolling your middle mouse wheel. Notice that the mouse location identifies the zooming center.

- 10) Fit all (**F**).

- 11) **Edit** your sketch. (See the informational block on **Editing a Sketch**.)



### Editing a sketch

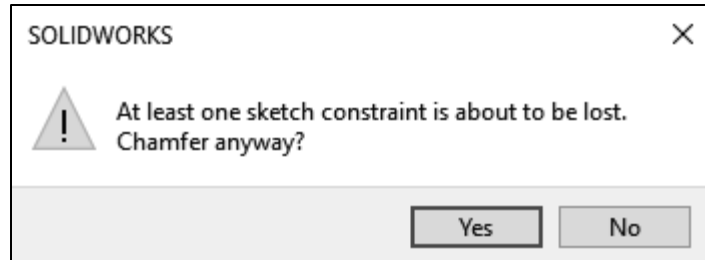
- 1) Select the sketch to be edited in the *Feature Manager Design Tree*.
- 2) Select **Edit Sketch**  from the *Context* toolbar.

The image shows a portion of the software interface. On the right is the 'Feature Manager Design Tree' with a list of features: 'connecting rod (Default<...>', 'History', 'Sensors', 'Annotations', 'Material <not specified>', 'Front Plane', 'Top Plane', and 'Sketch1'. A callout box labeled 'Edit sketch' points to the 'Edit Sketch' icon in the 'Context' toolbar, which is located below the 'Top Plane' and above 'Sketch1'. The 'Edit Sketch' icon is a pencil inside a square.

- 12) View your sketch from the normal plane by selecting **Ctrl + 8**.



13) Add two **2 x 45° Chamfers**



 **Sketch Chamfer** to the top outside corners and two **1 x 45° Chamfers** to the bottom outside corners. The *Chamfer* command is located under the *Fillet*  **Sketch Fillet** command.

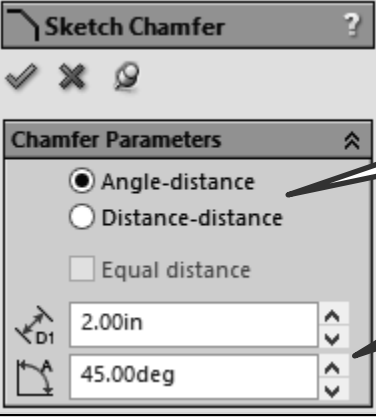


When applying the 1 x 45°, select **Yes** in the warning window. Note that this will delete the *Equal* relation between the two bottom horizontal lines. Reapply the **Equal** relation after you have applied the chamfer. (See the information block on **Sketch Chamfers & Sketch Fillets**.)



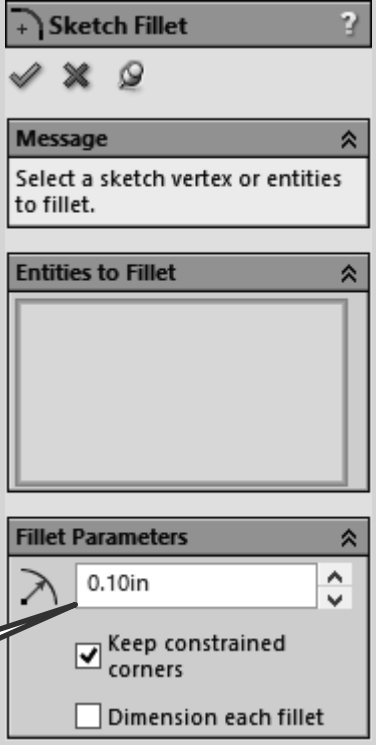
**Sketch Chamfers**  **Sketch Chamfer** & **Sketch Fillets**  **Sketch Fillet** (located in the *Sketch* tab)

- Chamfer:**  **Sketch Chamfer** A *Chamfer* is a beveled corner. It can be defined by selecting a vertex or two edges and then specifying a distance and an angle, or two distances.
- Fillet:**  **Sketch Fillet** A *Fillet* is a rounded corner. It is defined by selecting a vertex or two edges and then specifying a radius.



**Chamfer Parameters**

- ☒ Angle-distance
- ☐ Distance-distance
- ☐ Equal distance
- D1: 2.00in
- 45.00deg



**Fillet Parameters**

- 0.10in
- ☒ Keep constrained corners
- ☐ Dimension each fillet

**Note:** You should set your fillet or chamfer size before you select the entity.

**Note:** If a warning window appears, it means that by applying the chamfer or fillet you will be deleting a sketch relation or dimension. You may need to reapply the constraint.



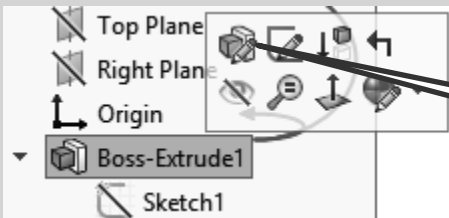
14) **Exit Sketch**

15) View your part from the isometric view (**Ctrl + 7**).

16) Change the **Extrude** distance from 2.5 to **2.1 inches**. (See the informational block on **Editing a Feature**.)

### Editing a feature

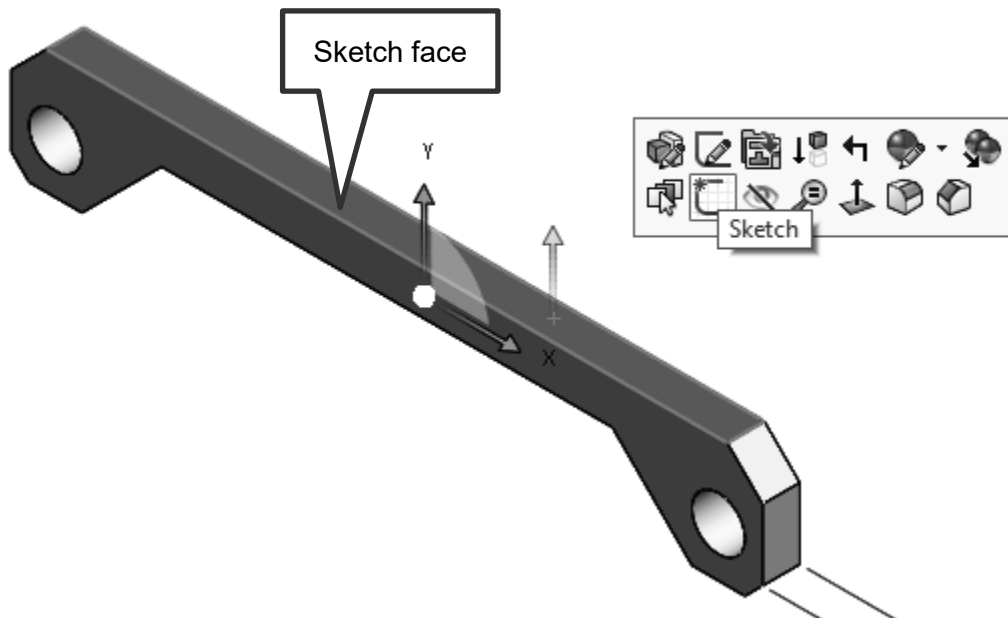
- 1) Select the feature to be edited in the *Feature Manager Design Tree*. Right clicking shows you the full command list.
- 2) Select **Edit Feature** from the *Context* toolbar.



Edit feature

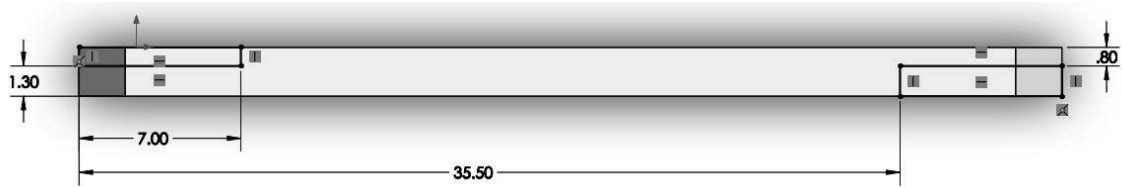
## 2.5) ADDING FEATURES

1) **Sketch** on the top face of your part.

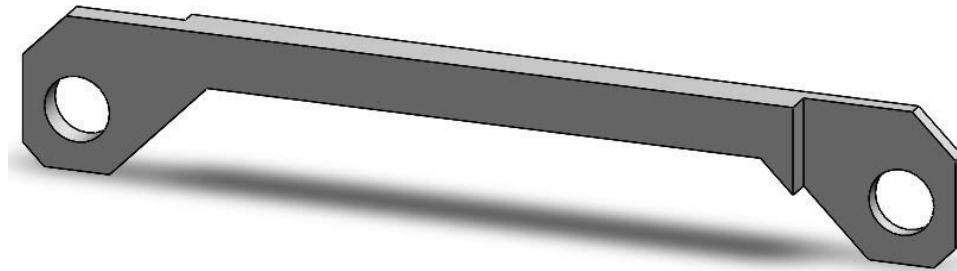


2) View the sketch from the normal direction (**Ctrl + 8**).

- 3) Sketch and dimension the following two **Rectangles**  **Corner Rectangle**. (See the informational block on **Rectangles**.)








- 4) **Extrude Cut** the rectangles **Through All**.



### **Rectangles** (located in the *Sketch* tab)

There are several ways to create a rectangle.

- 1) Corner Rectangle:  **Corner Rectangle** A *Corner Rectangle* is defined by two corner points.
- 2) Center Rectangle:  **Center Rectangle** A *Center Rectangle* is defined by a geometric center and a corner point.
- 3) 3 Point Corner Rectangle:  **3 Point Corner Rectangle** A *3 Point Corner Rectangle* is defined by three corner points. This allows you to specify the rectangle's orientation.
- 4) 3 Point Center Rectangle:  **3 Point Center Rectangle** A *3 Point Center Rectangle* is defined by a center point, a corner point, and a side midpoint. This allows you to specify the rectangle's orientation.
- 5) Parallelogram:  **Parallelogram** A *Parallelogram* is defined by three corner points.

- 5) Add **R1.00 Fillets** to the area where the angled lines meet the main body of the rod. You may need to **rotate** your part to view the underside of the part. (See the informational blocks on **Rotating your part** and **Fillets & Chamfers**.)



### Rotating your part

#### Method 1

- 1) Click and hold your **middle mouse button**.
- 2) Move your mouse.

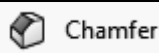
#### Method 2

- 1) Use the *View Selector* (**Space bar**).

### Fillets




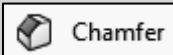
### & Chamfers

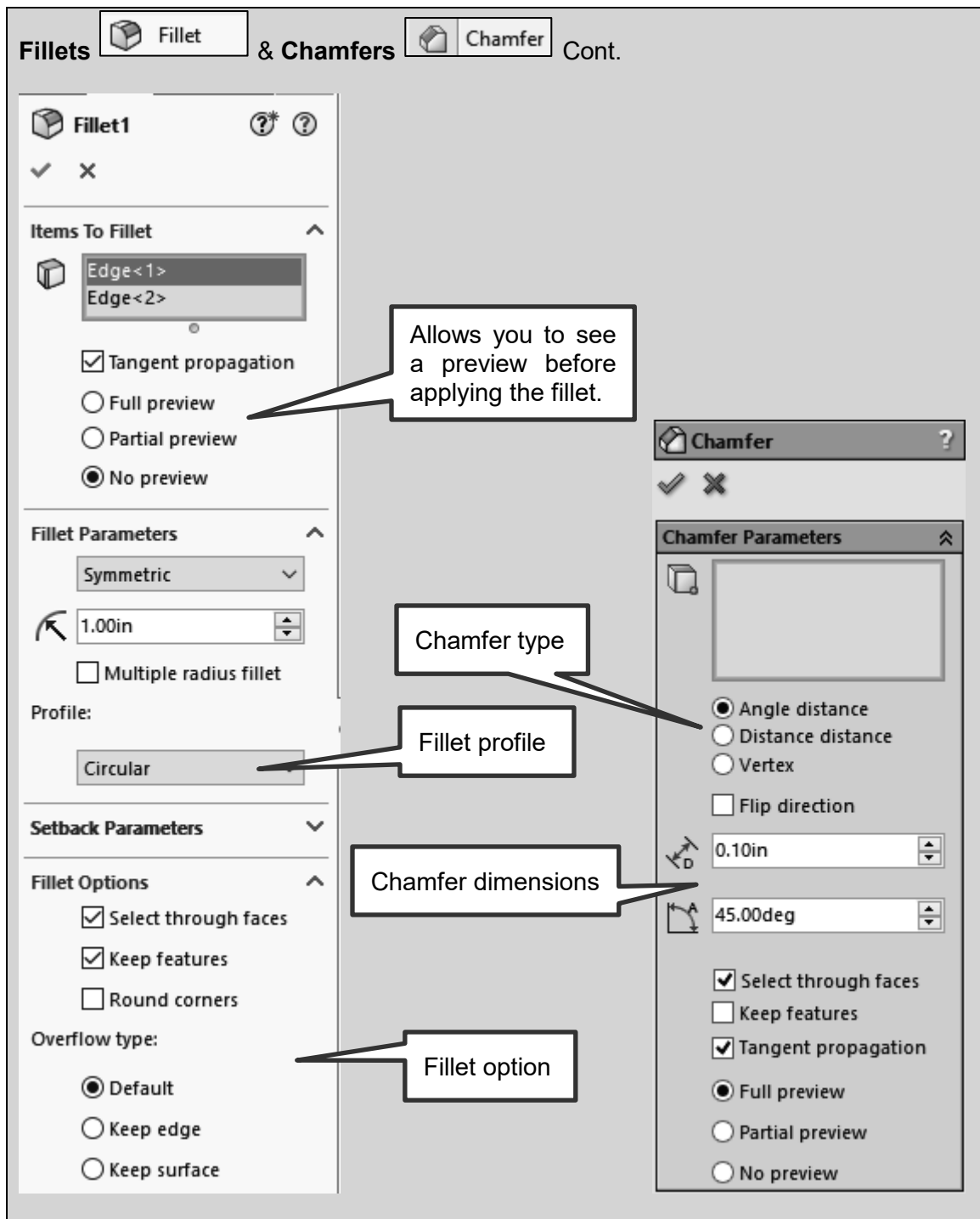


(located in the *Features* tab)

The *Fillet* and *Chamfer* commands are similar to the *Sketch Fillet* and *Sketch Chamfer* commands previously described. The difference is that these commands are applied to a solid and not a sketch.

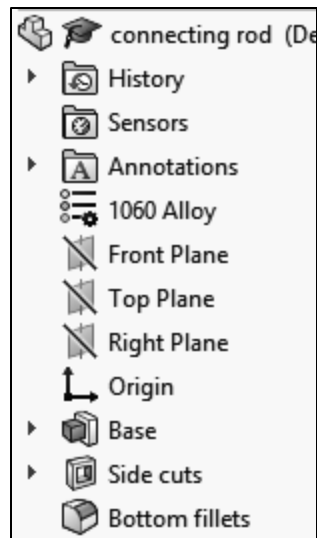
- 1) **Fillet:**  A *Fillet* is a rounded corner. It is created by selecting two faces or an edge and then specifying its radius. You may apply several fillets at once. You can also specify the fillet type and profile.

- 2) **Chamfer:**  A *Chamfer* is a beveled corner. It is created by selecting two faces or an edge and then specifying either a distance and angle or two distances. You may apply several chamfers at once.



6) In your *Feature Manager Design Tree*, name your features as shown. To name your feature, slowly double click on the name.

7) **Save**

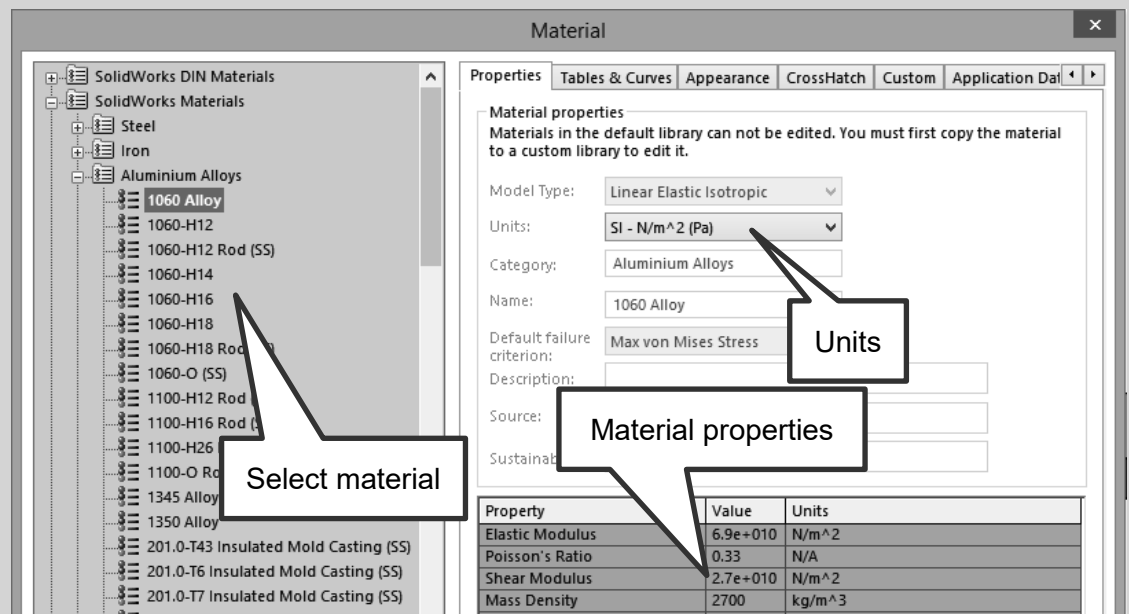
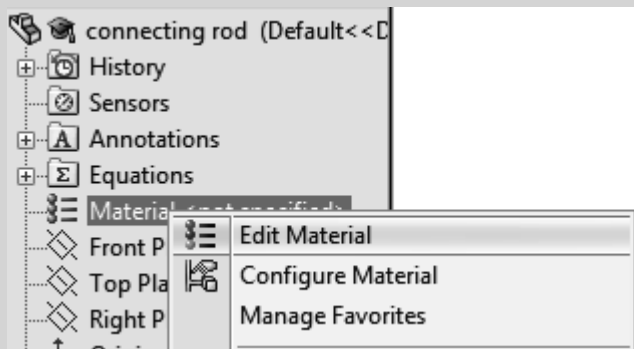


## 2.6) ADDING MATERIAL

1) Apply a material of **Aluminum 1060 Alloy** to your part.  
(See the informational block on **Applying Material**.)

### Applying material

- 1) Right click on **Material** in the *Feature Manager Design Tree*.
- 2) Select **Edit Material**.
- 3) A *Material* window will appear.
- 4) Select your desired material.
- 5) Select **Apply**.
- 6) Select **Close**.





- 2) Calculate the weight of your part. In the **Evaluate** tab, select **Mass Properties**

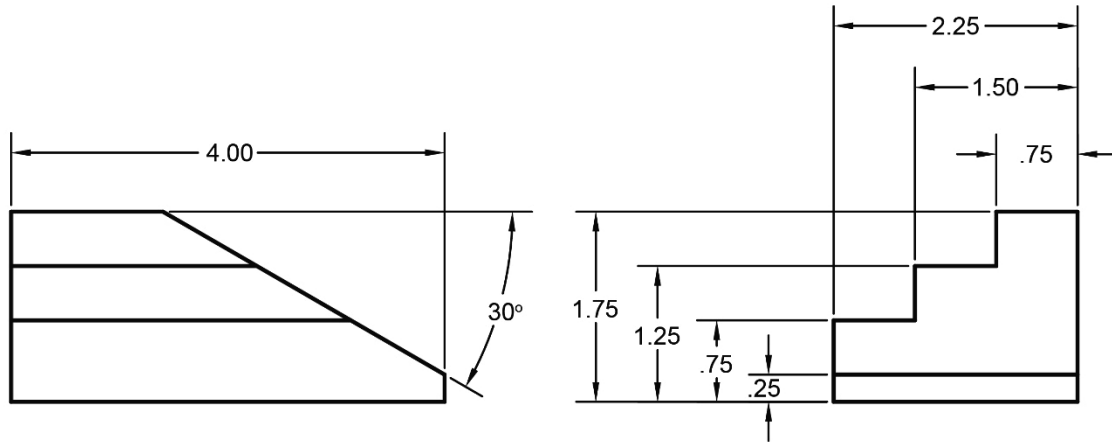


. In the *Mass Properties* window note that the mass of your part is 22.22 lb. This is really the weight of your part because of the units. If your weight is not this value, your model is incorrect. This window also gives other physical properties.

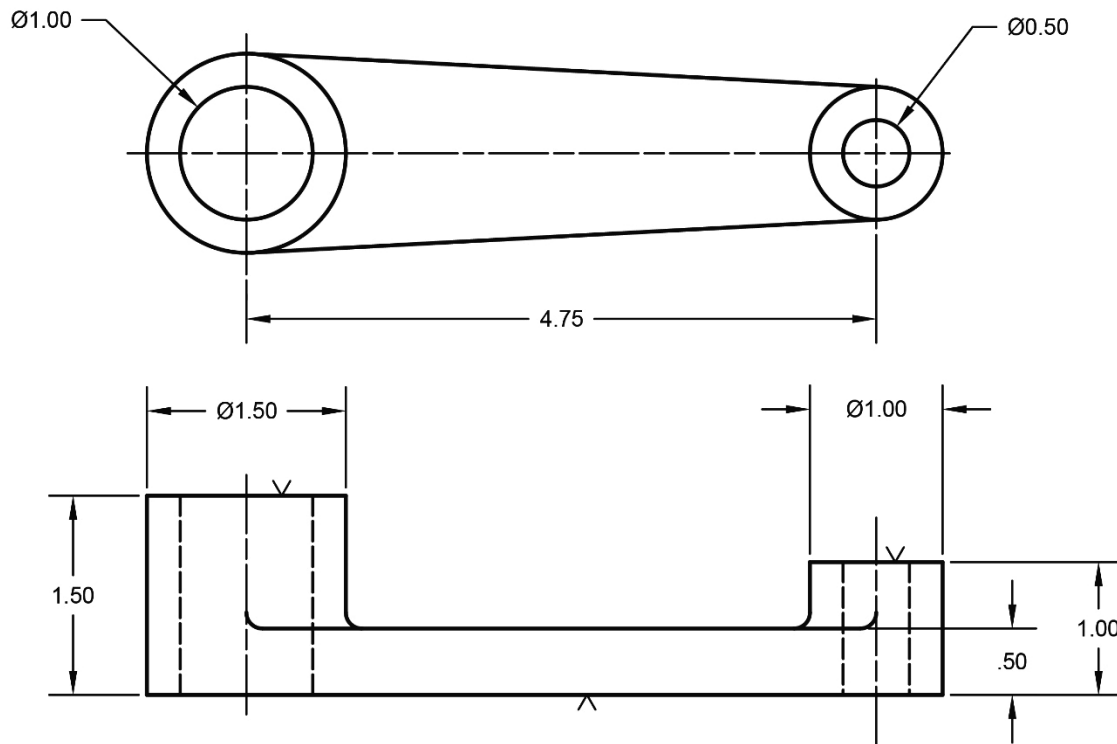
NOTES:

## CONNECTING ROD PROJECT (MODEL) PROBLEMS

**P2-1)** Create a solid model of the following 1345 Aluminum part and calculate the weight of the part. Dimensions are given in inches.

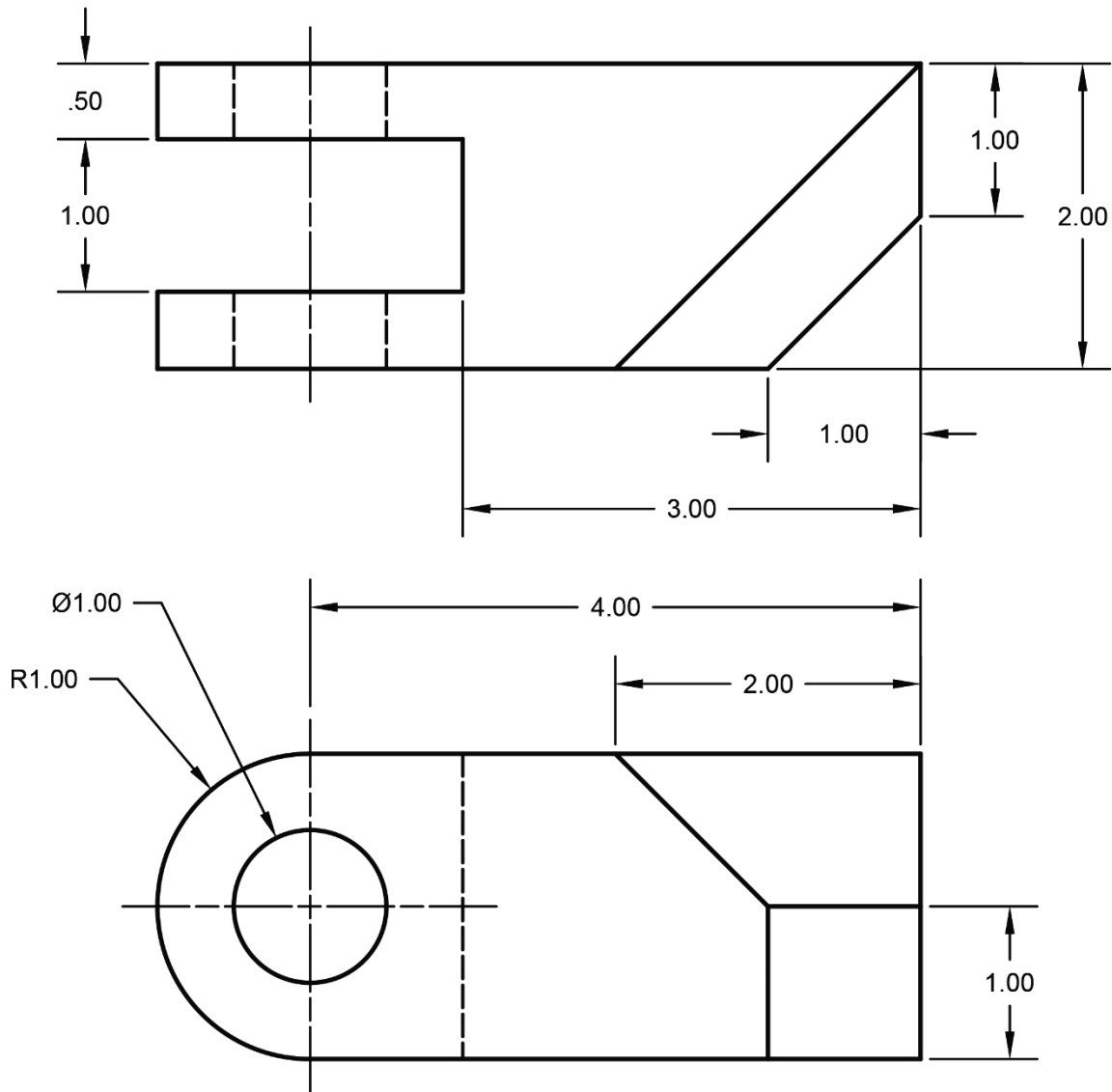


**P2-2)** Create a solid model of the following Gray Cast Iron part and calculate the weight of the part. Dimensions are given in inches.

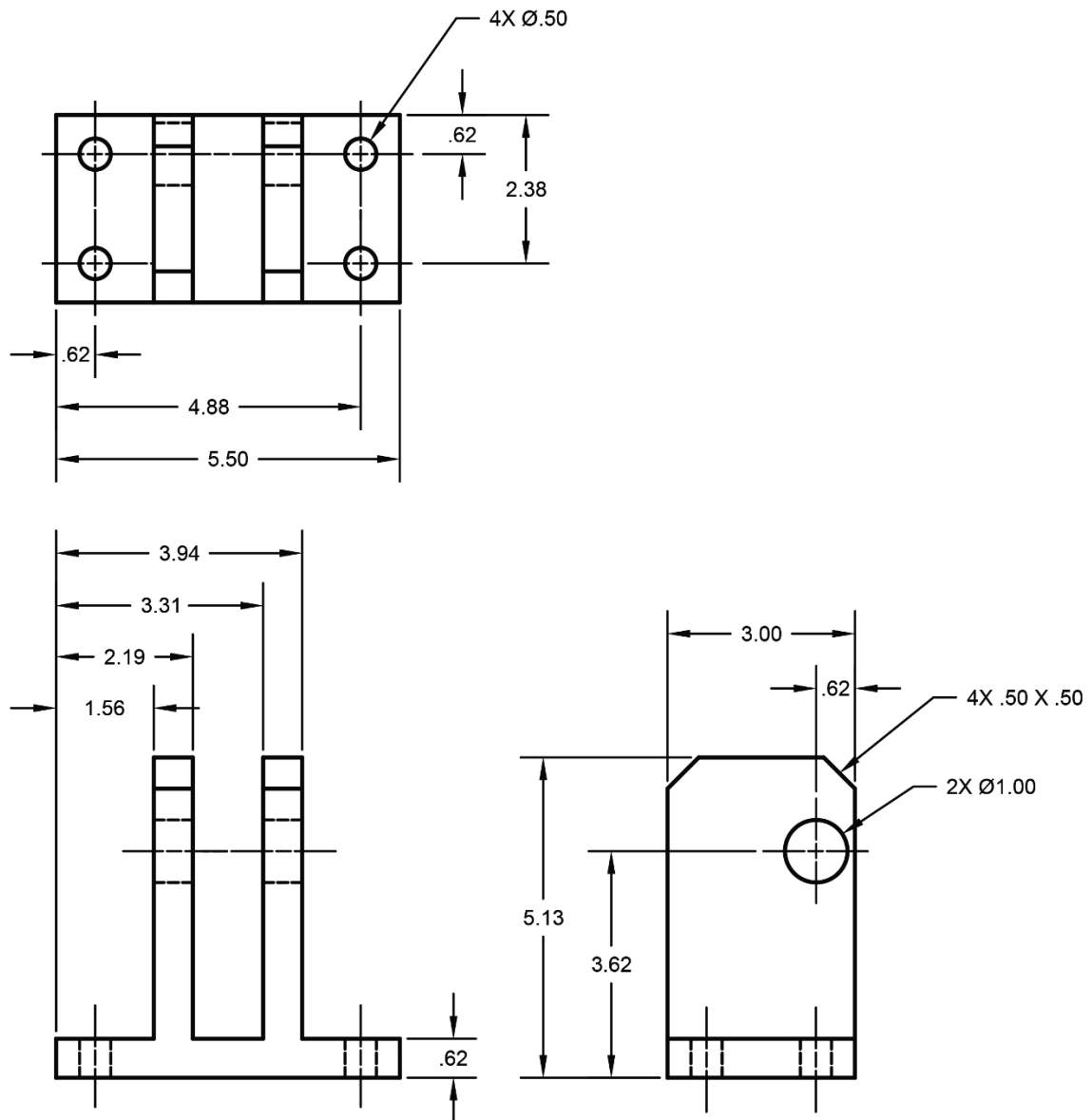


NOTE: ALL FILLETS AND ROUNDS R.12  
UNLESS OTHERWISE SPECIFIED

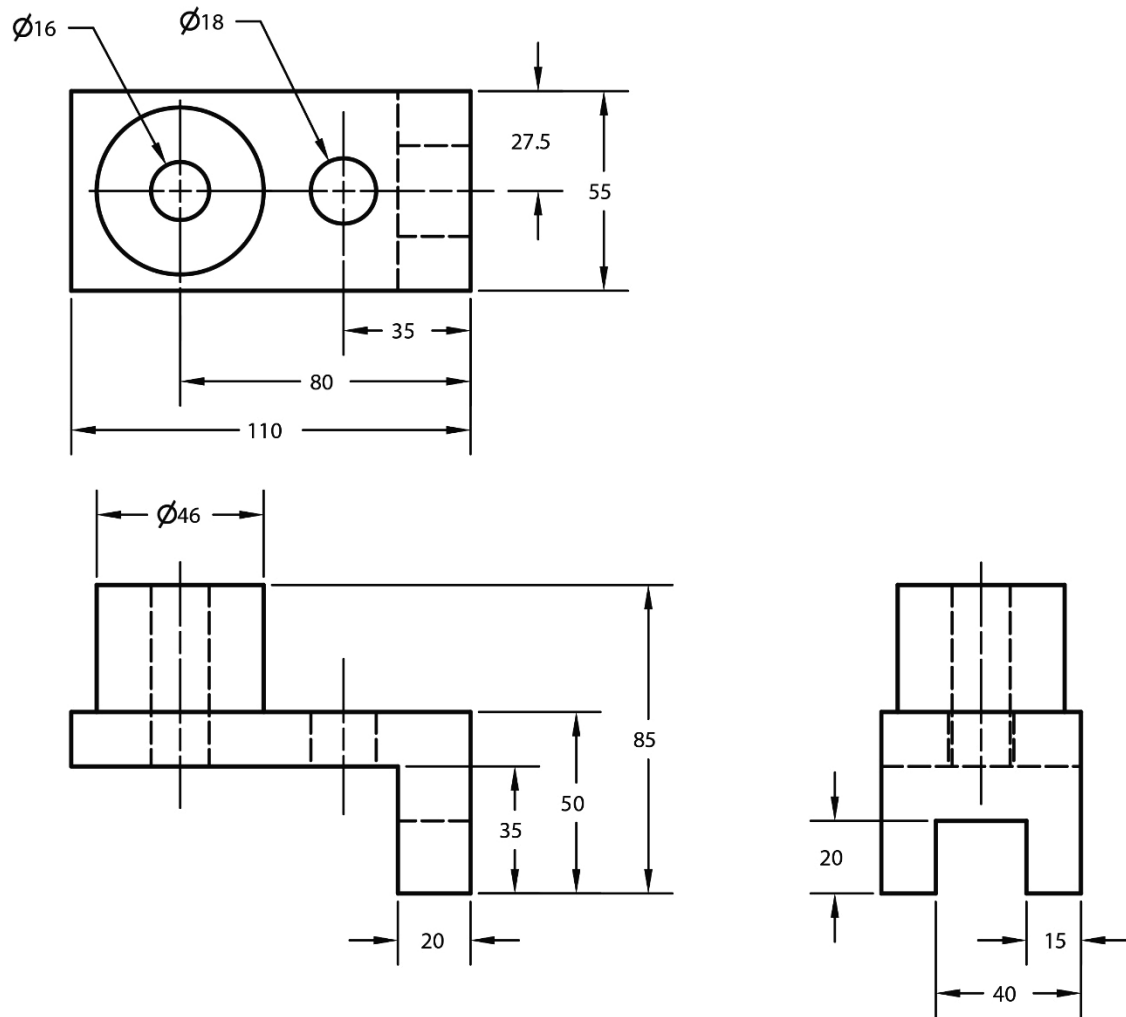
**P2-3)** Create a solid model of the following 1020 Steel part and calculate the weight of the part. Dimensions are given in inches.



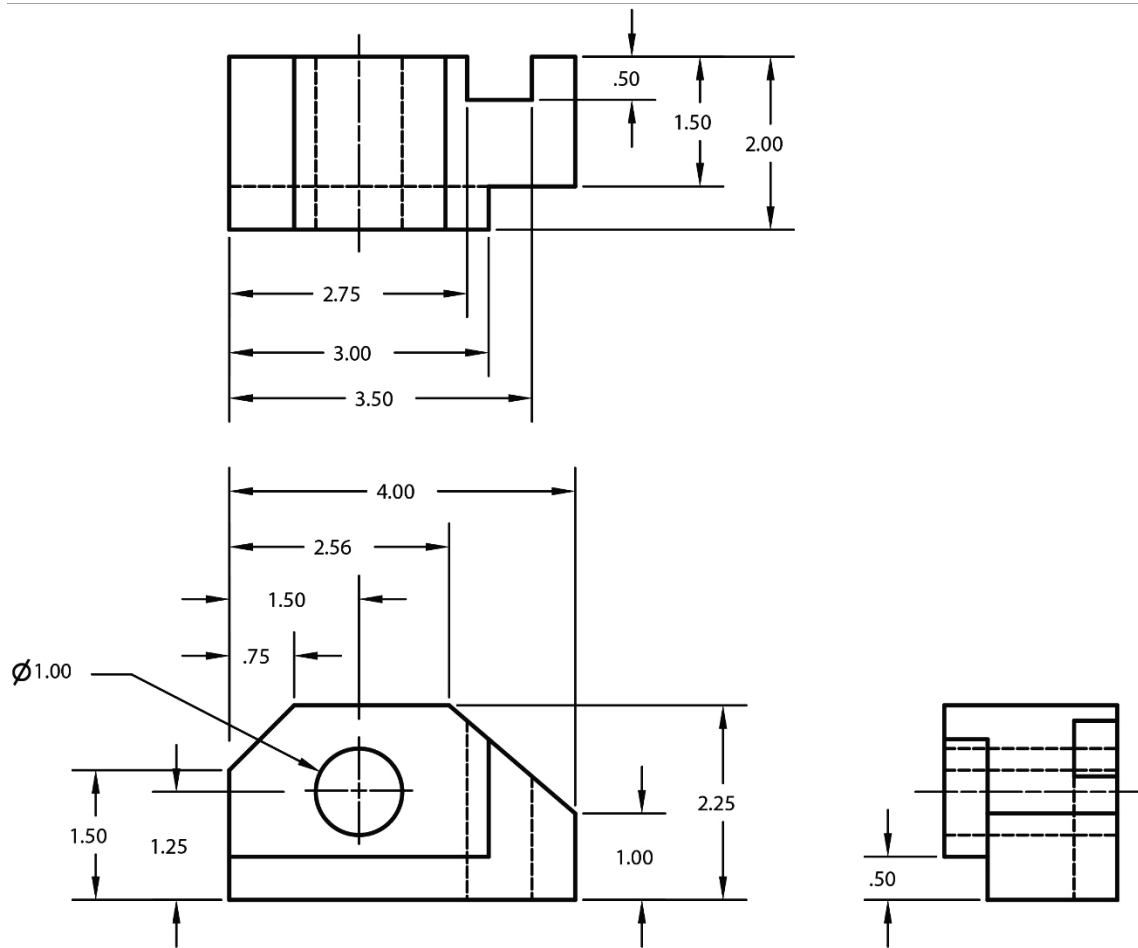
**P2-4)** Create a solid model of the following ABS plastic part and calculate the weight of the part. Dimensions are given in inches.



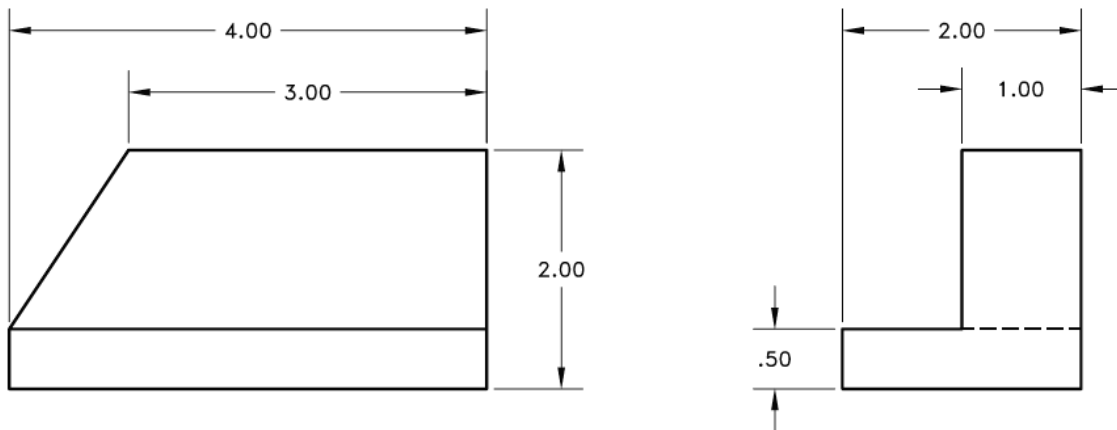
**P2-5)** Create a solid model of the following Brass part and calculate the mass of the part. Dimensions are given in millimeters.



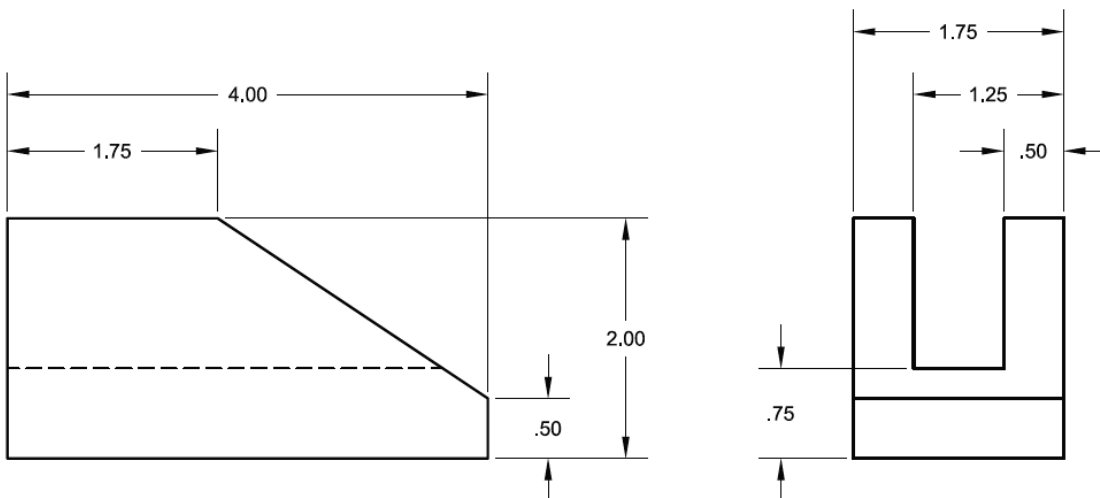
**P2-6)** Create a solid model of the following Oak part and calculate the weight of the part. Dimensions are given in inches.



**P2-7)** Use SOLIDWORKS® to create a solid model of the following 1345 Aluminum part. Calculate the weight of your part. Dimensions are given in inches.

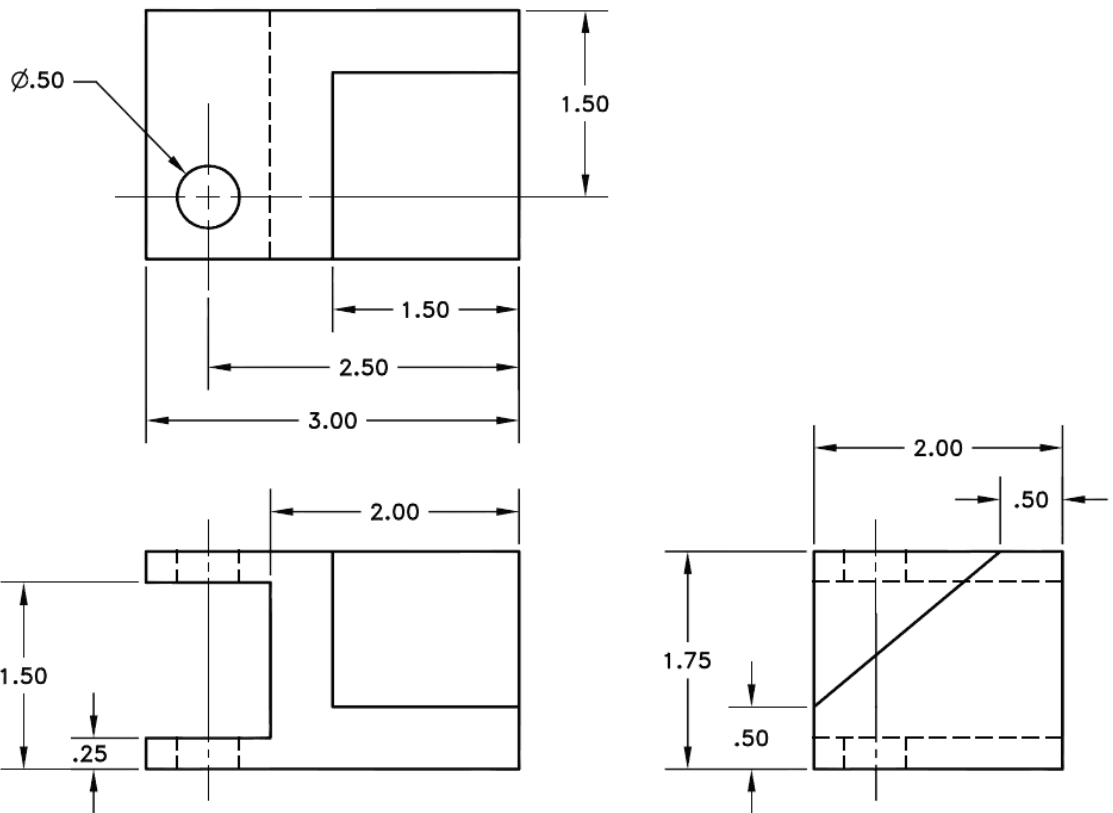


**P2-8)** Use SOLIDWORKS® to create a solid model of the following 6061 Aluminum part. Calculate the weight of your part. Dimensions given in inches.

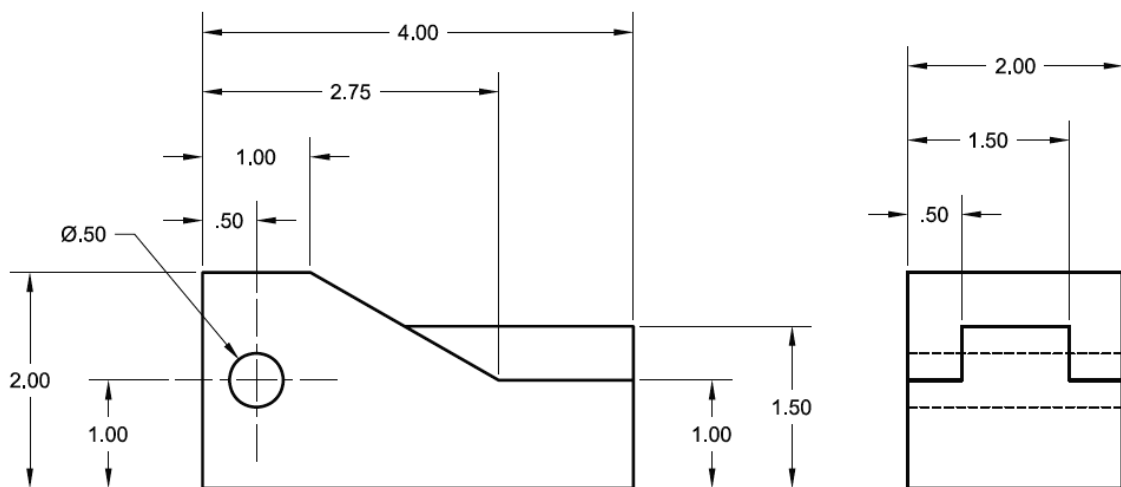




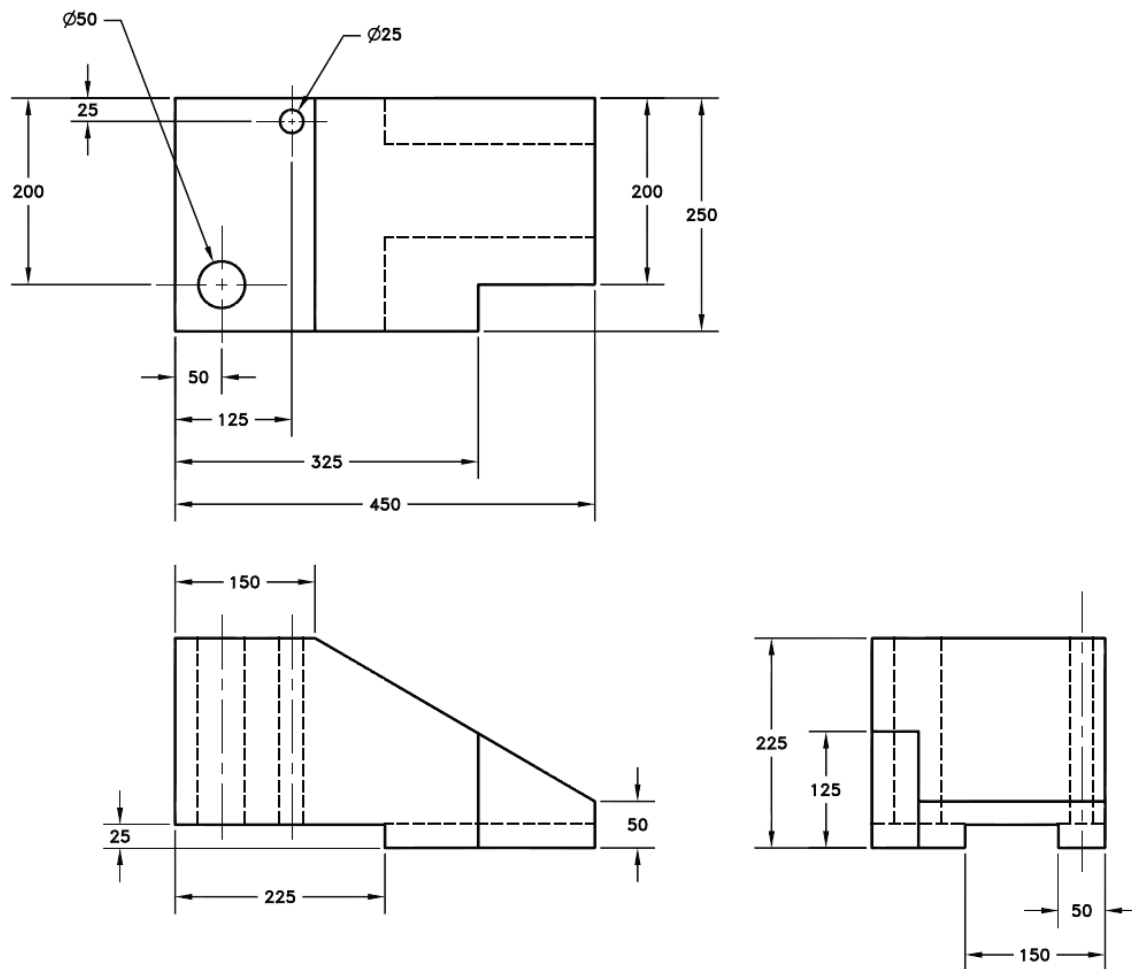
**P2-9)** Use SOLIDWORKS® to create a solid model of the following 1020 Steel part. Calculate the weight of your part. Dimensions given in inches.



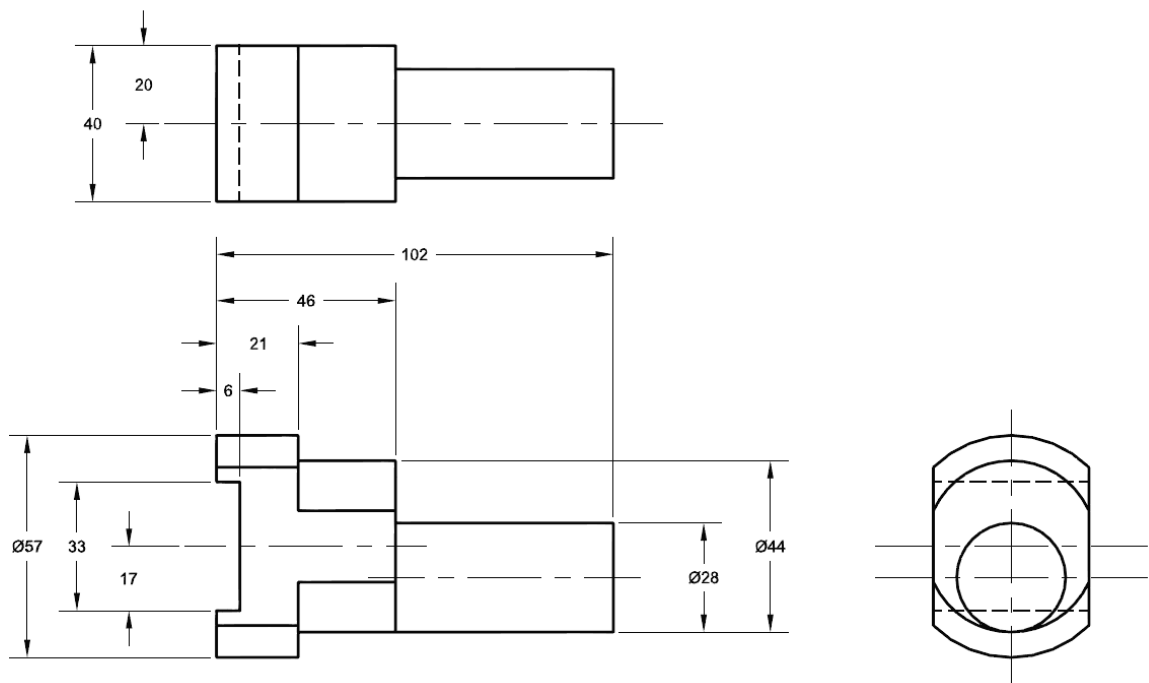
**P2-10)** Use SOLIDWORKS® to create a solid model of the following 1020 Steel part. Calculate the weight of your part. Dimensions given in inches.



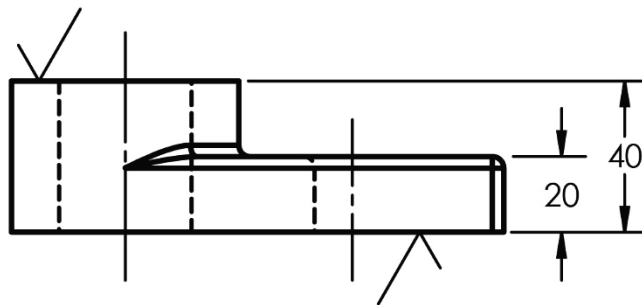
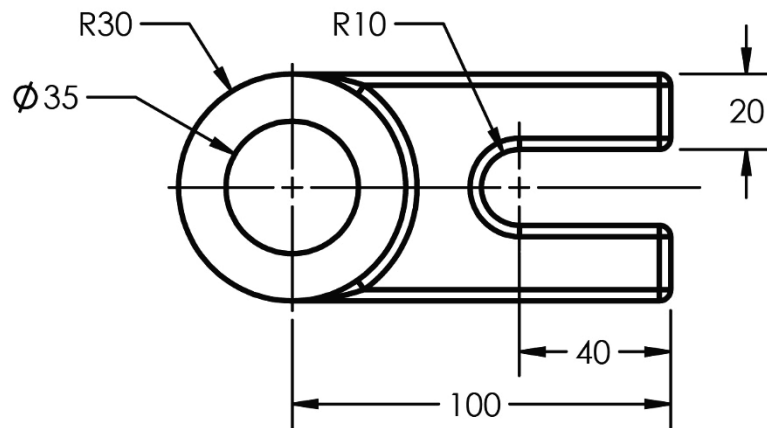
**P2-11)** Use SOLIDWORKS® to create a solid model of the following ABS plastic part. Calculate the weight of your part. Dimensions given in millimeters.



**P2-12)** Use SOLIDWORKS® to create a solid model of the following Oak part. Calculate the mass of your part. Dimensions given in millimeters.



**P2-13)** Use SOLIDWORKS® to create a solid model of the following Grey Cast Iron. Calculate the mass of your part. Dimensions given in millimeters. Note that all fillets and rounds are R3.



## CONNECTING ROD PROJECT (MODEL) QUIZ PROBLEMS

**Q2-1)** Use SOLIDWORKS® to create a solid model of the following 1060 Aluminum Alloy part. Dimensions given in millimeters.

a) Calculate the mass of your part and circle the correct answer.

- 1835.02 grams
- 1724.08 grams
- 1040.73 grams
- 998.01 grams
- 783.99 grams
- 726.04 grams

Group 1

A = 100

B = 90

C = 50

Group 2

A = 110

B = 100

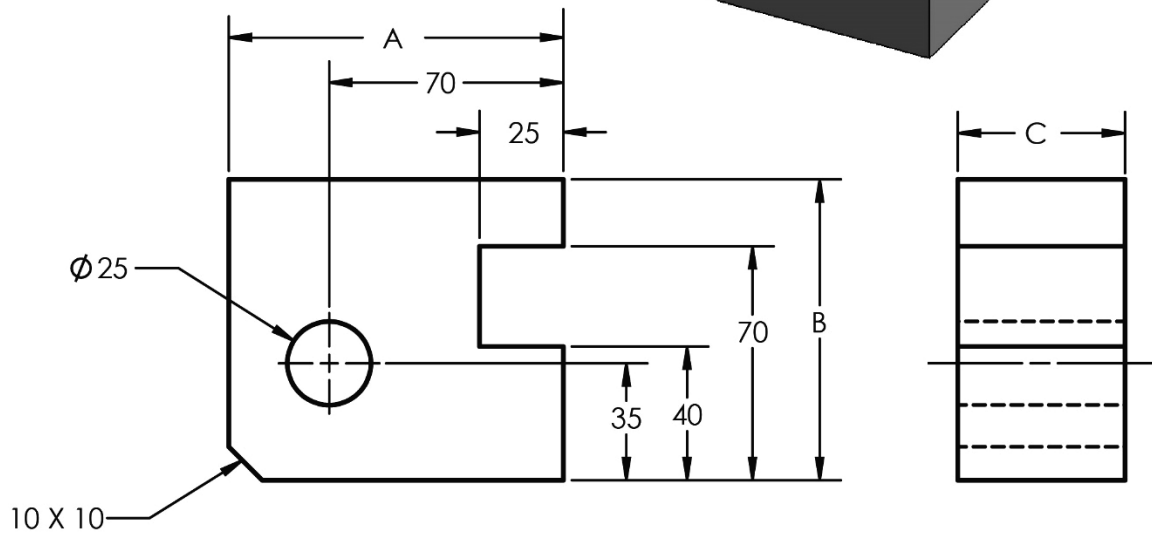
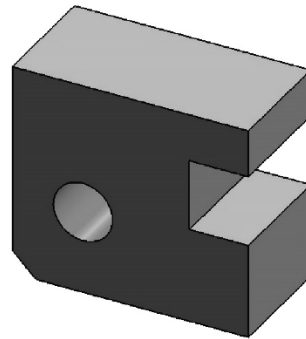
C = 70

Group 3

A = 90

B = 95

C = 40



- b) Make the following modifications to your part and calculate the mass.
- Add three R10 fillets as shown.
  - Cut a hexagon through on the right side of the part as shown.

Mass = \_\_\_\_\_ grams

