

# Project Based SOLIDWORKS® 2021

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## CHAPTER 2

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# CONNECTING ROD PROJECT Model

### CHAPTER OUTLINE

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## 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

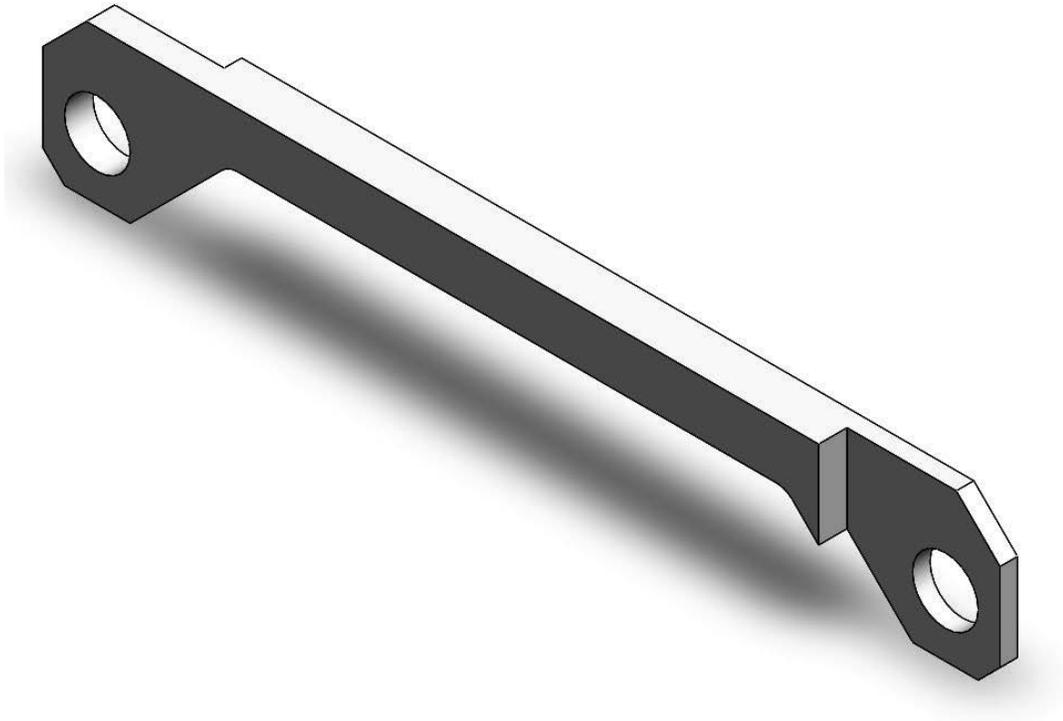
- Chamfer
- Fillets
- Editing a feature

### Material and properties

- Applying material
- Mass properties

### View

- Panning
- Rotating

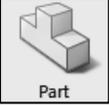


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Figure 2.2-1: Connecting rod

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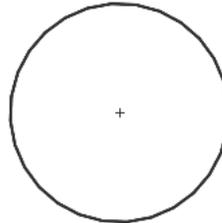
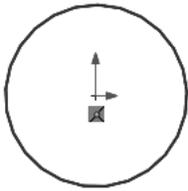
### 2.3) SETTING UP THE PROJECT

- 1) **Start SOLIDWORKS** and then start a new  **part** .
- 2) Set your drafting standard to **ANSI** and set all your text to **upper case** for notes, tables and dimensions. (Options  – *Document Properties – Drafting Standard*)
- 3) Set your unit to **IPS** (i.e. inch, pound, second) and set your **Decimals = .12**. Also, select the rounding option, **Round half to even**. (Options  – *Document Properties – Units*)
- 4) Save your part as **CONNECTING ROD.SLDPRT** (**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**

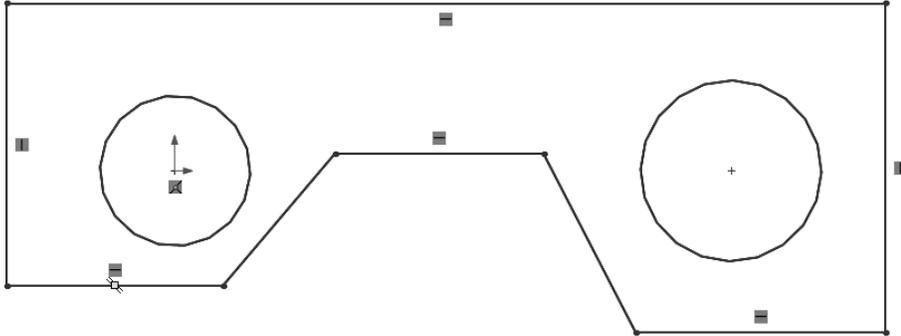
#### Method 1

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

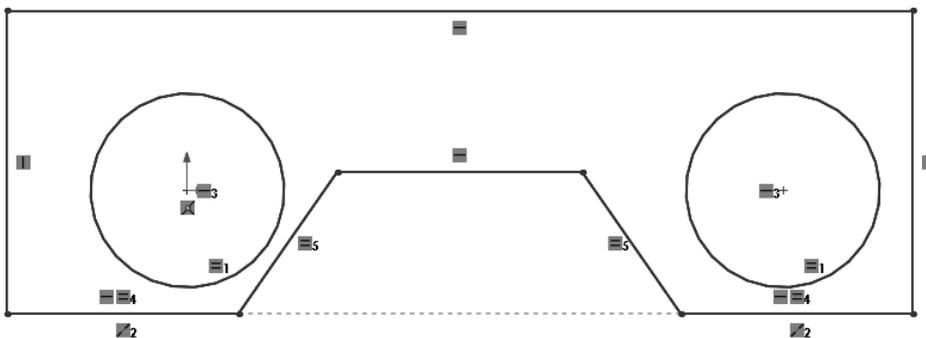
#### Method 2

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

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



### Viewing relations

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

## Applying sketch relations

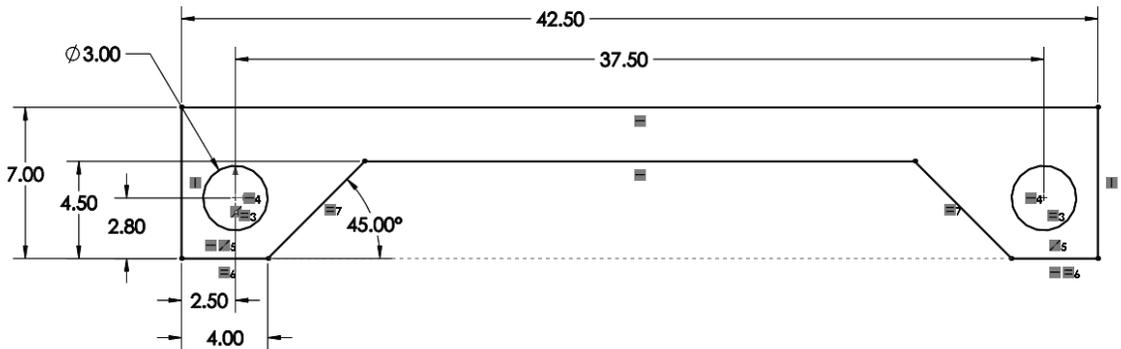
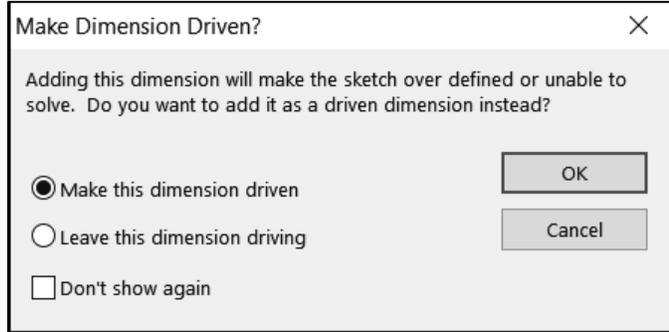
Sketch relations add geometric constraints between two or more entities. For example, we can make two lines parallel, or two circles concentric.

- 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.





- 7) Add the **Dimensions** shown. 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: 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*.)

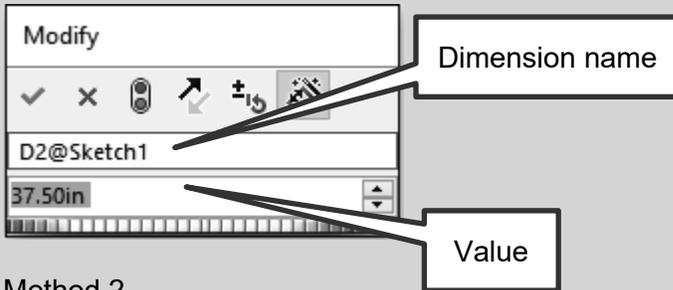


- 8) After you are done dimensioning, you should make the left circle and the origin **Coincident**. Note that there should be **no blue lines** when you are finished. Blue lines mean that it is under-constrained.

## Editing a dimension

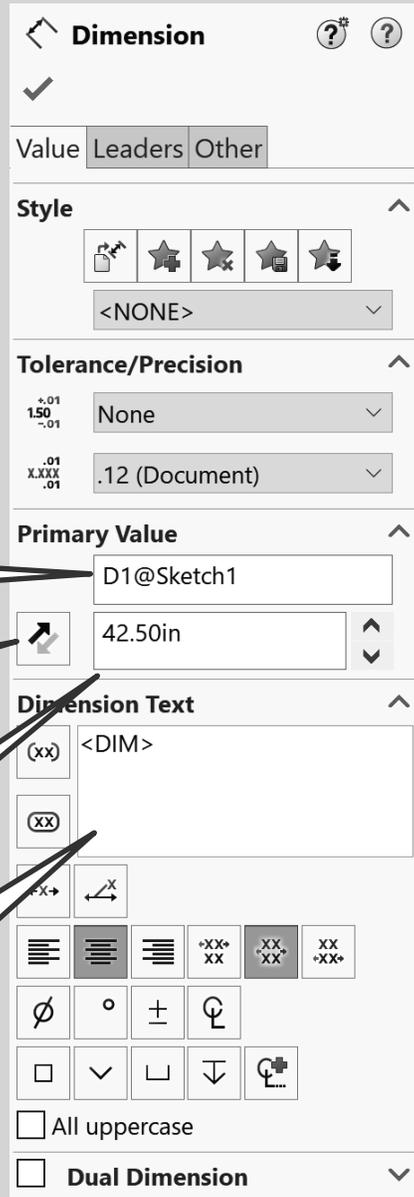
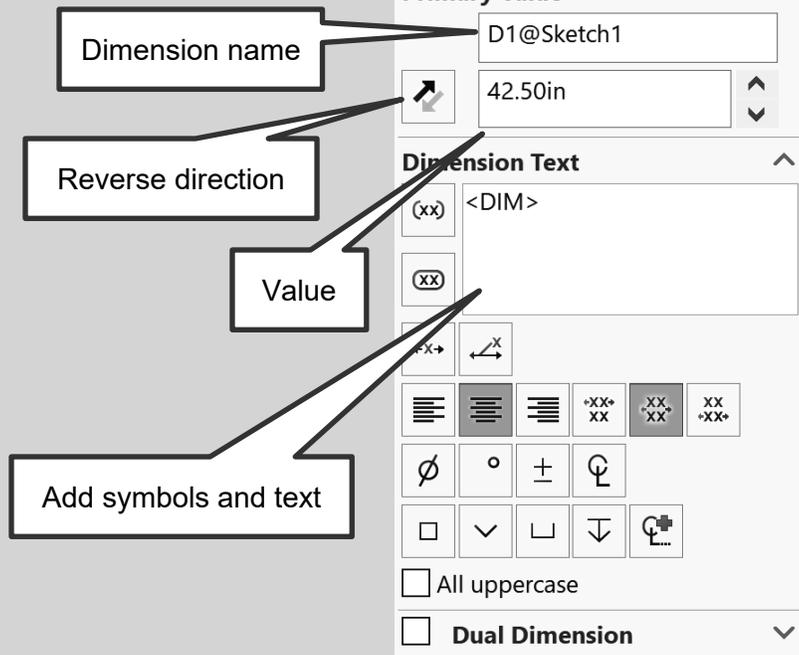
### Method 1

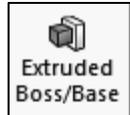
- 1) Double click the dimension that you wish to edit.
- 2) Within the *Modify* window, fill in the correct dimension value and then select the green check mark .



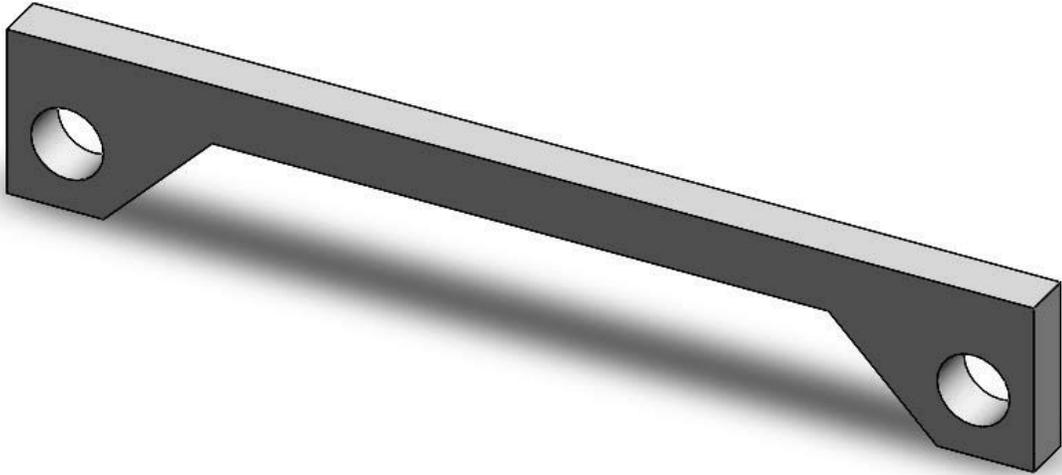
### Method 2

- 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.





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



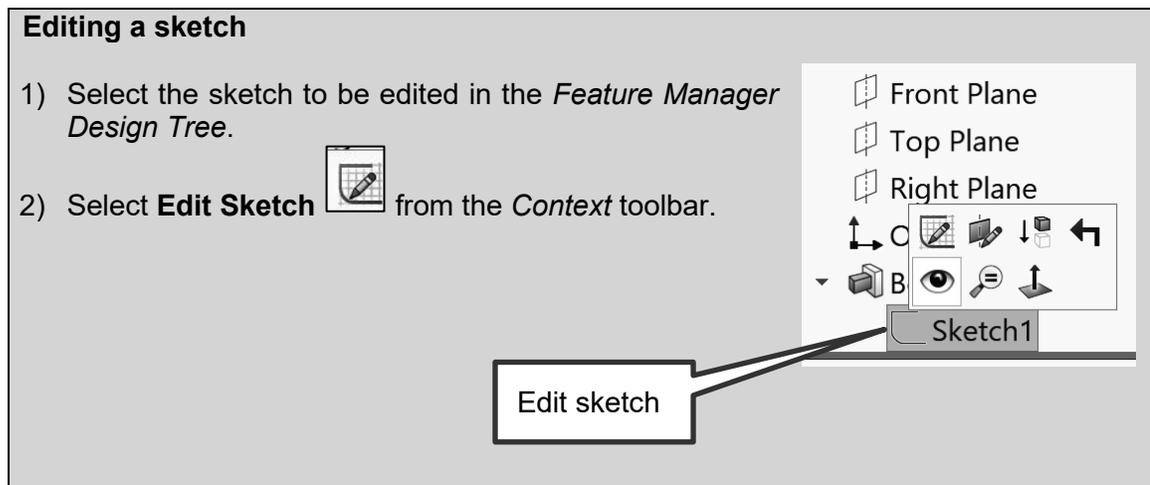
10) Try **zooming in and out** by scrolling your middle mouse wheel. Notice that the mouse location identifies the zooming center.

11) Fit all (**F**).

12) **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.



Front Plane  
Top Plane  
Right Plane  
C  
B  
Sketch1

Edit sketch

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

14) Add two **2 x 45° Chamfers**

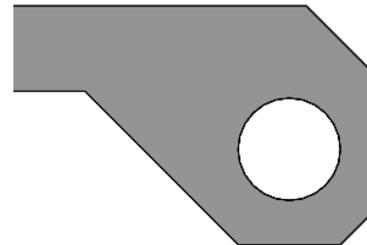
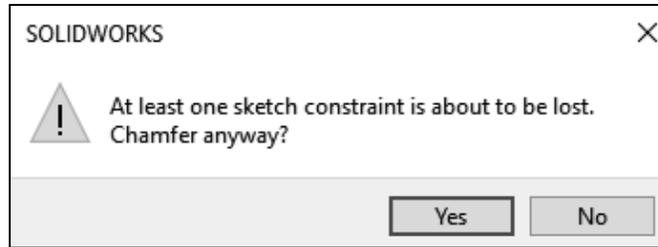


to the top outside corners and two **1 x 45° Chamfers** to the bottom outside corners. The *Chamfer* command is located under the

*Fillet*



command. (See the information block on **Sketch Chamfers & Sketch Fillets**.) 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. After you reapply the Equal relation, you may be asked to **Rebuild and Save**.



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

- 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:** 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
- Distance: 2.00in
- Angle: 45.00deg

**Fillet Parameters**

- Radius: 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.



15) **Exit Sketch**

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

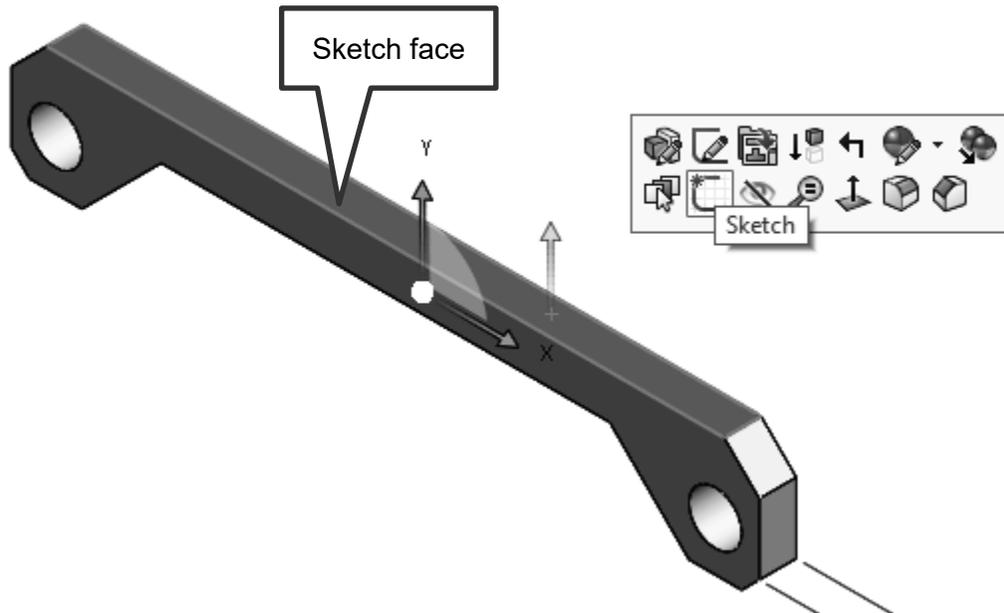
17) Change the **Extrude** distance from 2.5 to **2.1 inches**. (See the informational block on **Editing a Feature**.)

**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.

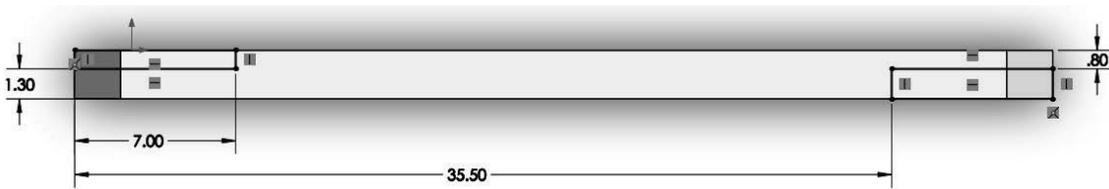
**2.5) ADDING FEATURES**

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

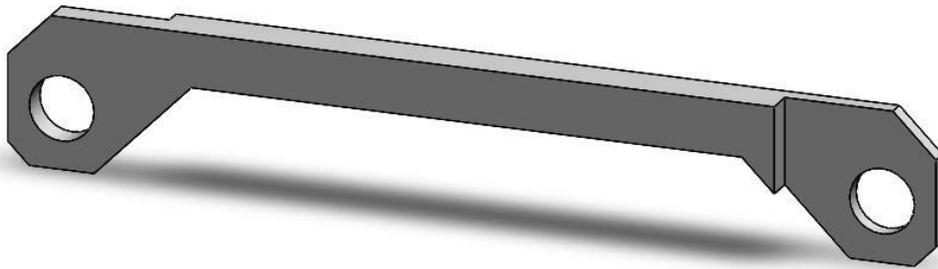


2) View the sketch from the normal direction (**Ctrl + 8**).

- 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.

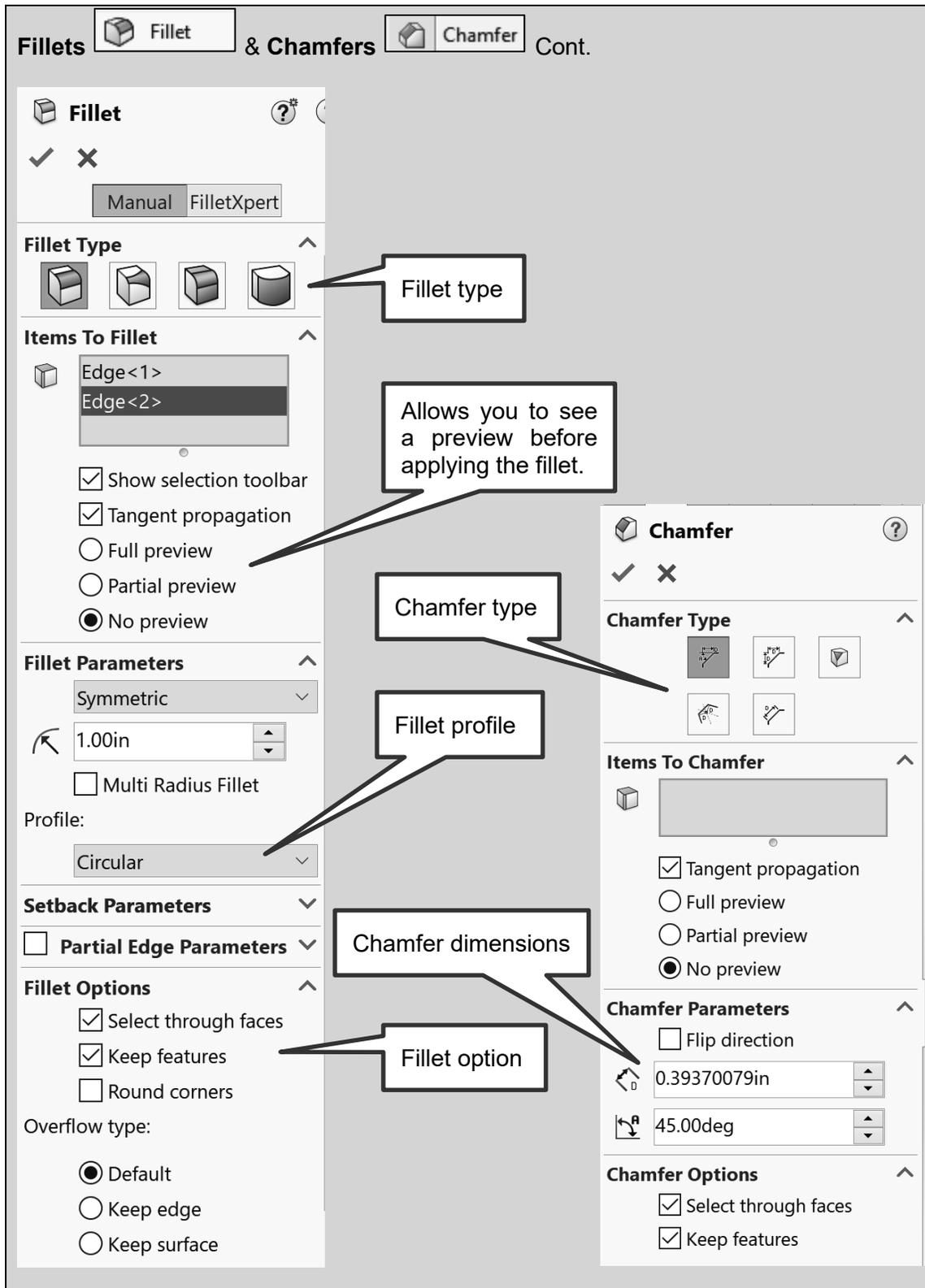
#### Method 2

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

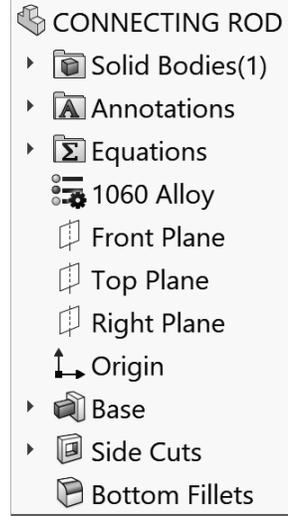
### Fillets & Chamfers (located in the *Features* tab)

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.**

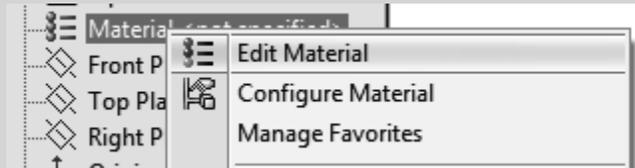


## 2.6) ADDING MATERIAL

- 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**.



**Material**

Properties | Tables & Curves | Appearance | CrossHatch | Custom | Application Def...

Material properties  
Materials in the default library can not be edited. You must first copy the material to a custom library to edit it.

Model Type: Linear Elastic Isotropic

Units: SI - N/m<sup>2</sup> (Pa)

Category: Aluminium Alloys

Name: 1060 Alloy

Default failure criterion: Max von Mises Stress

Description:

Source:

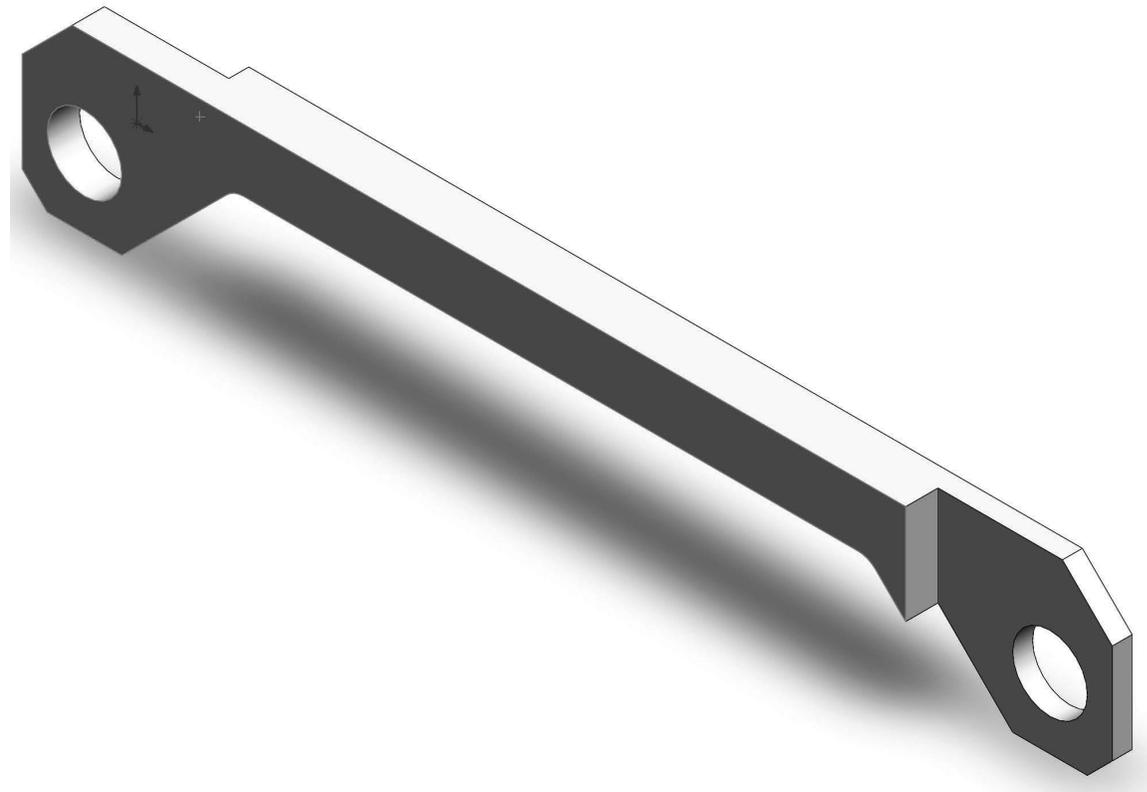
Sustainable:

Property	Value	Units
Elastic Modulus	6.9e+010	N/m <sup>2</sup>
Poisson's Ratio	0.33	N/A
Shear Modulus	2.7e+010	N/m <sup>2</sup>
Mass Density	2700	kg/m <sup>3</sup>

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

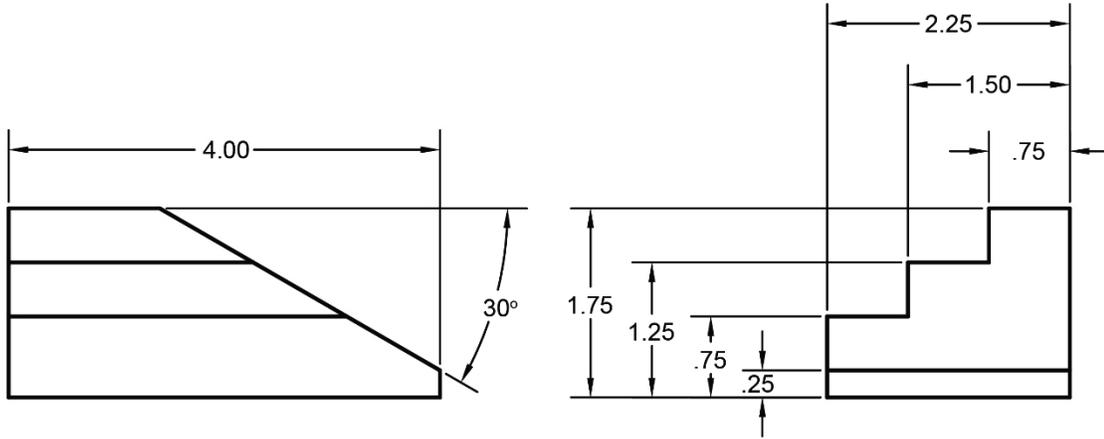


. In the *Mass Properties* window note that the mass of your part is 22.22 lb. This is really the weight of your part because of the units. If your weight is not this value, your model is incorrect. This window also gives other physical properties.

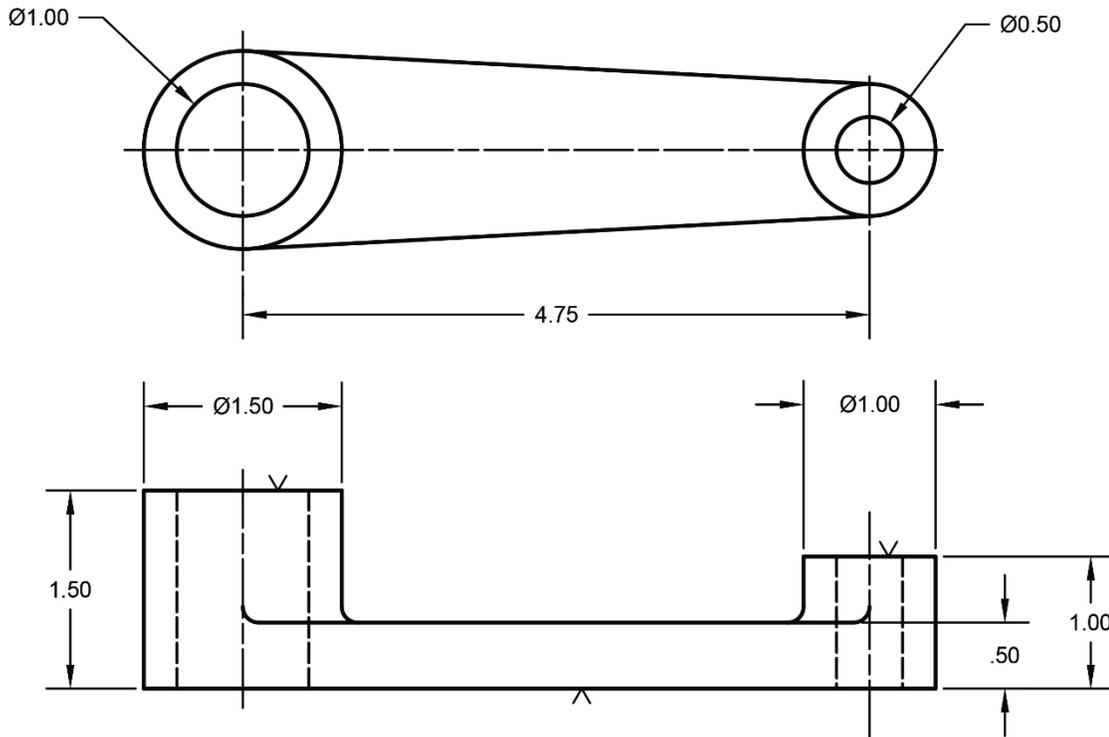


## CONNECTING ROD PROJECT (MODEL) PROBLEMS

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

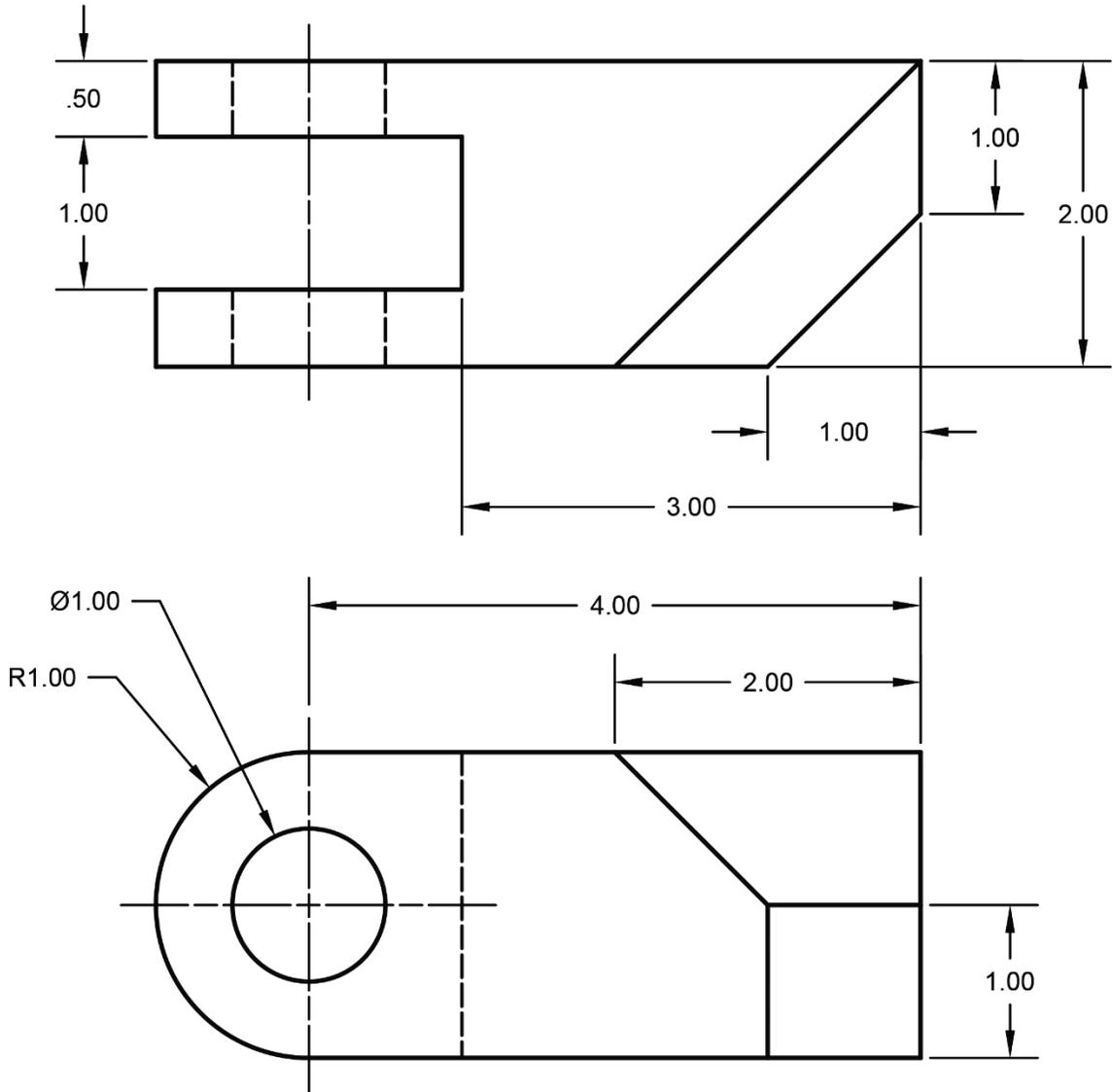


**P2-2)** Create a solid model of the following Gray Cast Iron part and calculate the weight of the part. Dimensions are given in inches.

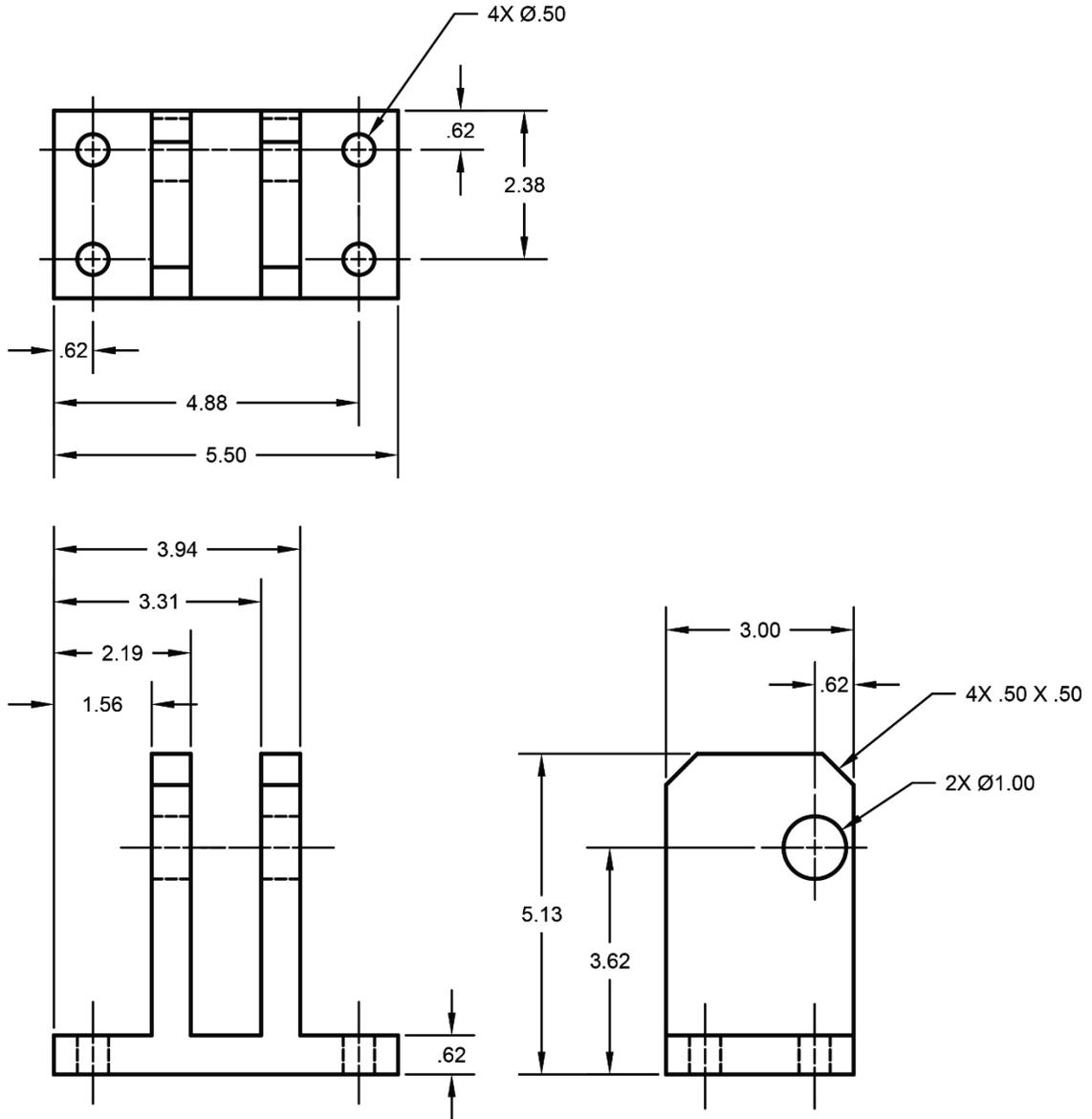


NOTE: ALL FILLETS AND ROUNDS R.12  
UNLESS OTHERWISE SPECIFIED

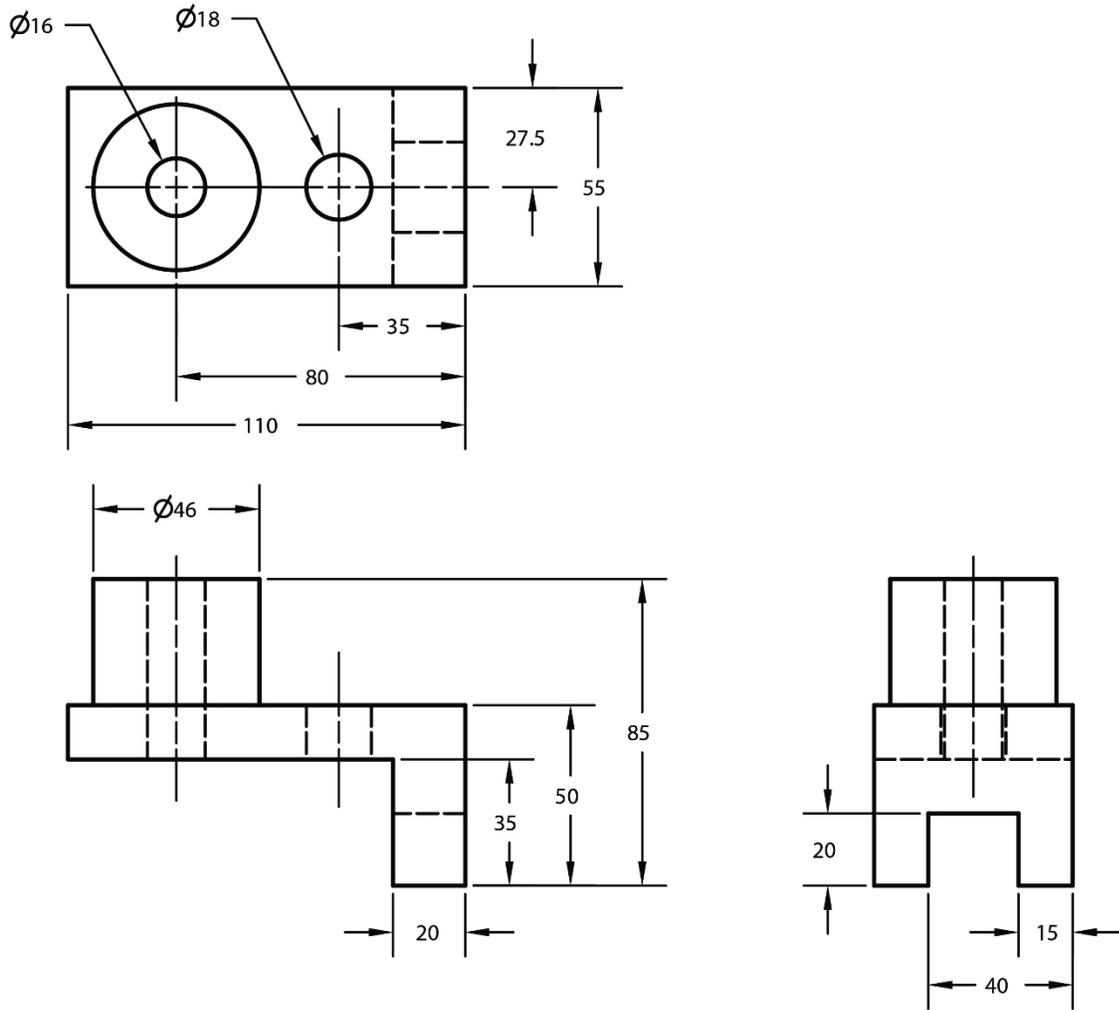
**P2-3)** Create a solid model of the following 1020 Steel part and calculate the weight of the part. Dimensions are given in inches.



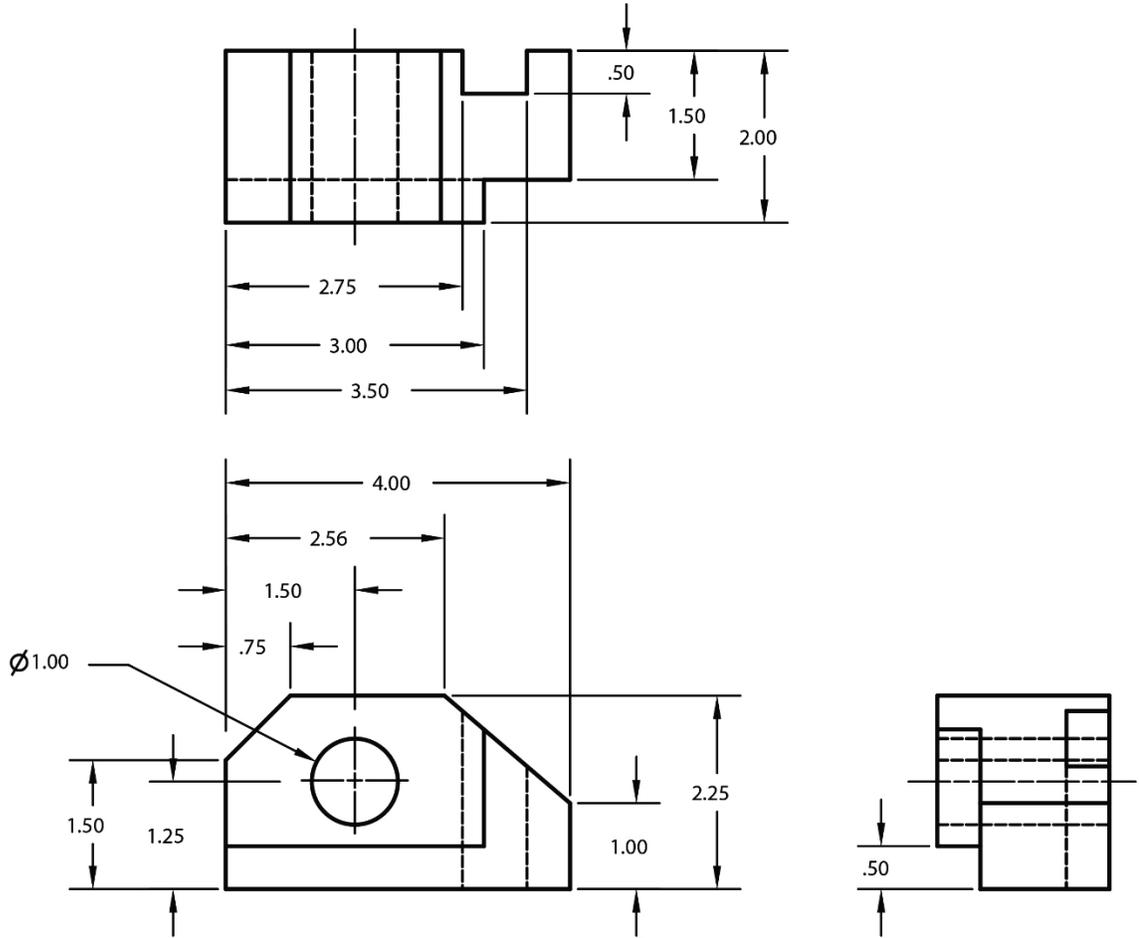
**P2-4)** Create a solid model of the following ABS plastic part and calculate the weight of the part. Dimensions are given in inches.



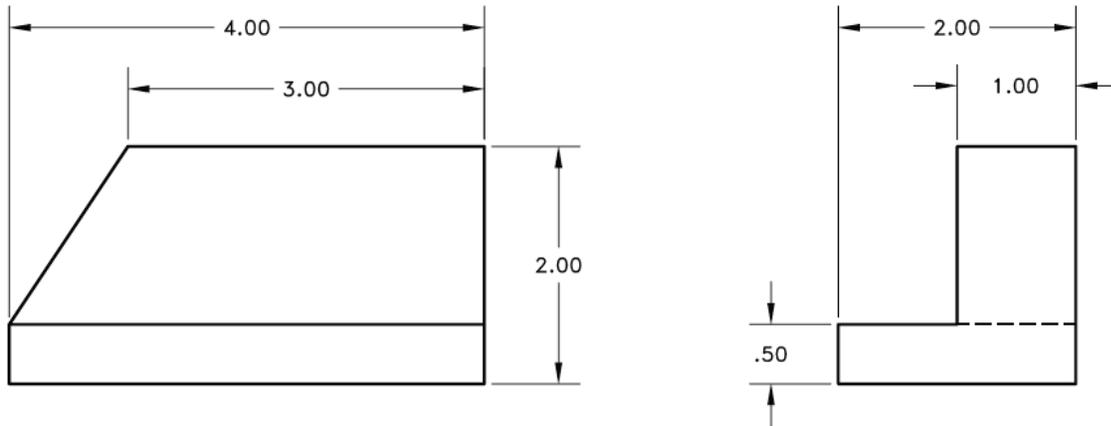
**P2-5)** Create a solid model of the following Brass part and calculate the mass of the part. Dimensions are given in millimeters.



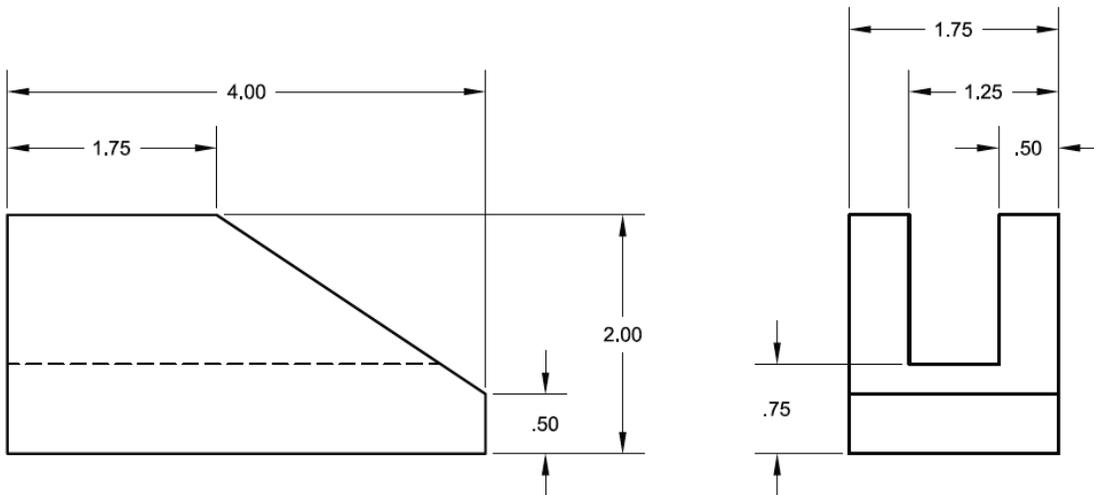
**P2-6)** Create a solid model of the following Oak part and calculate the weight of the part. Dimensions are given in inches.



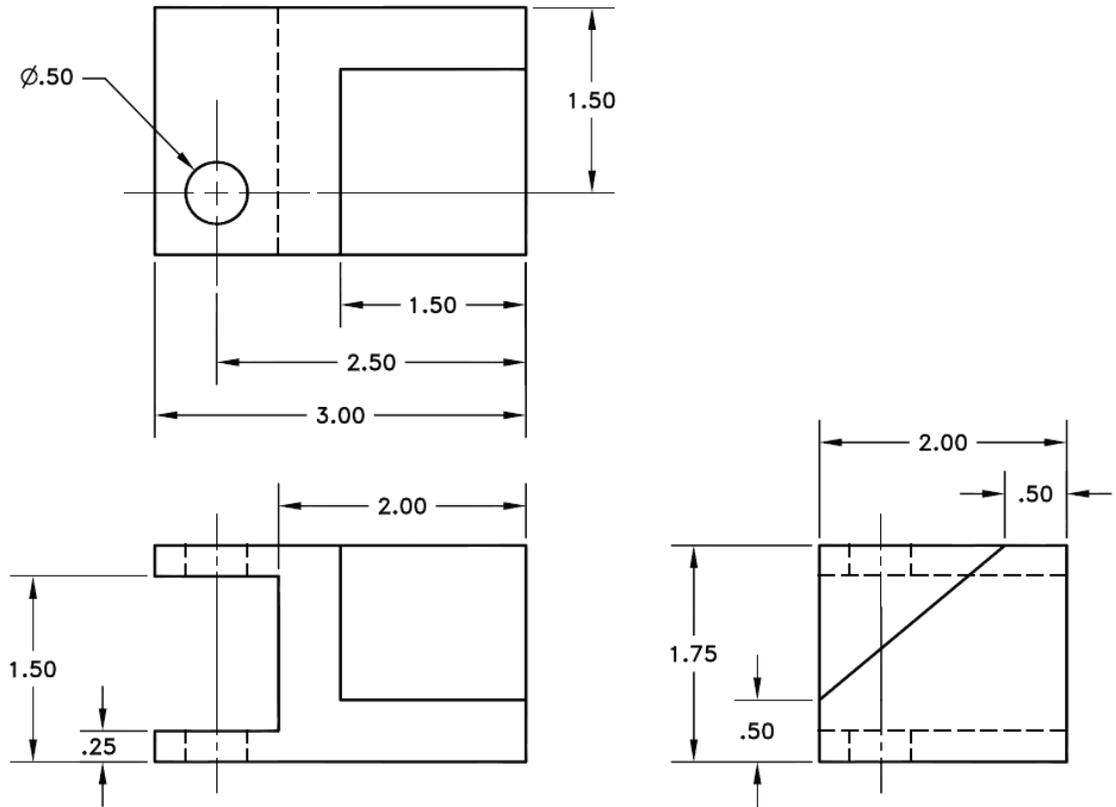
**P2-7)** Use SOLIDWORKS® to create a solid model of the following 1345 Aluminum part. Calculate the weight of your part. Dimensions are given in inches.



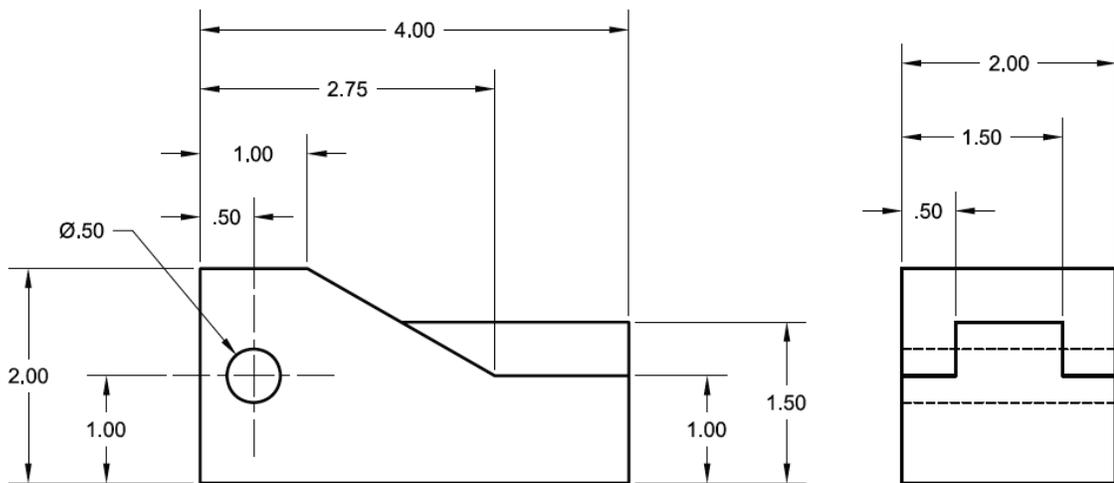
**P2-8)** Use SOLIDWORKS® to create a solid model of the following 6061 Aluminum part. Calculate the weight of your part. Dimensions given in inches.



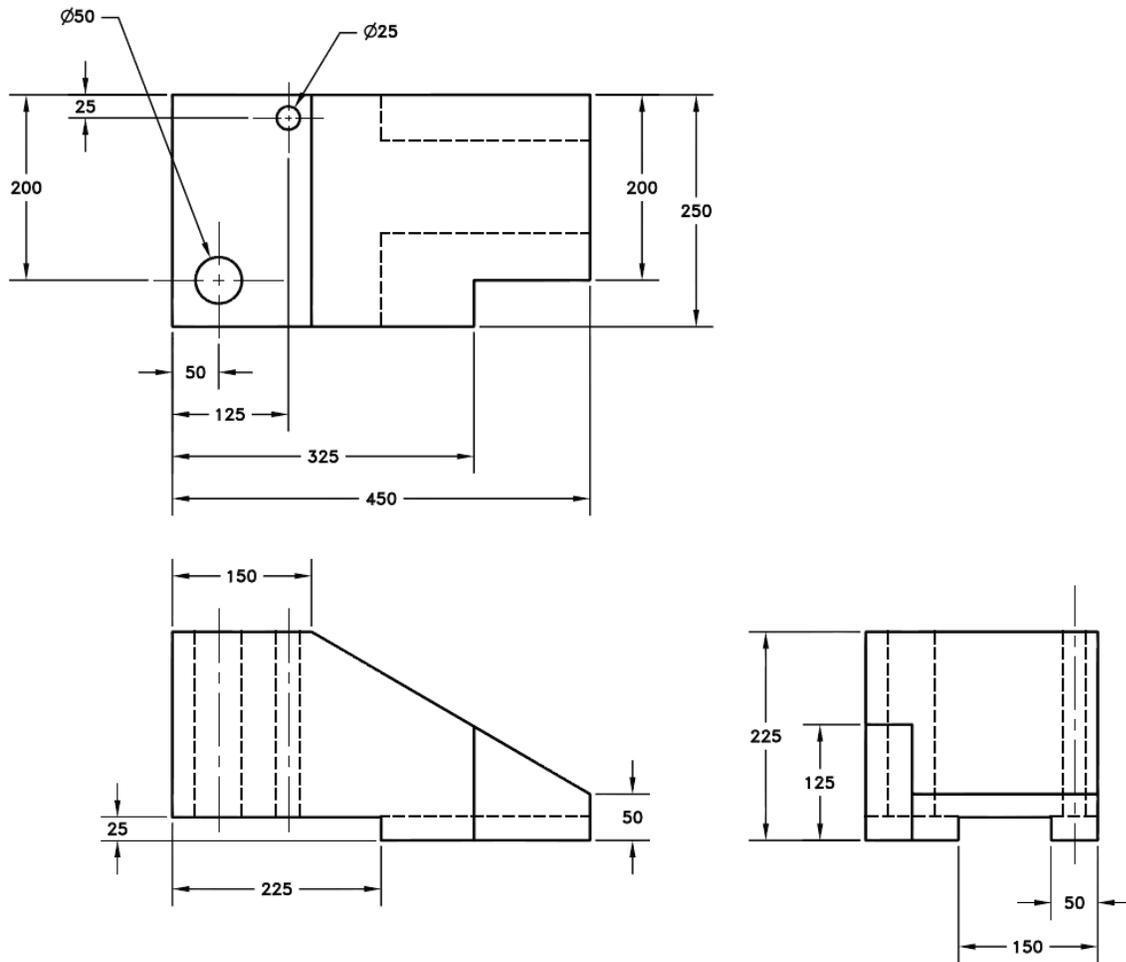
**P2-9)** Use SOLIDWORKS® to create a solid model of the following 1020 Steel part. Calculate the weight of your part. Dimensions given in inches.



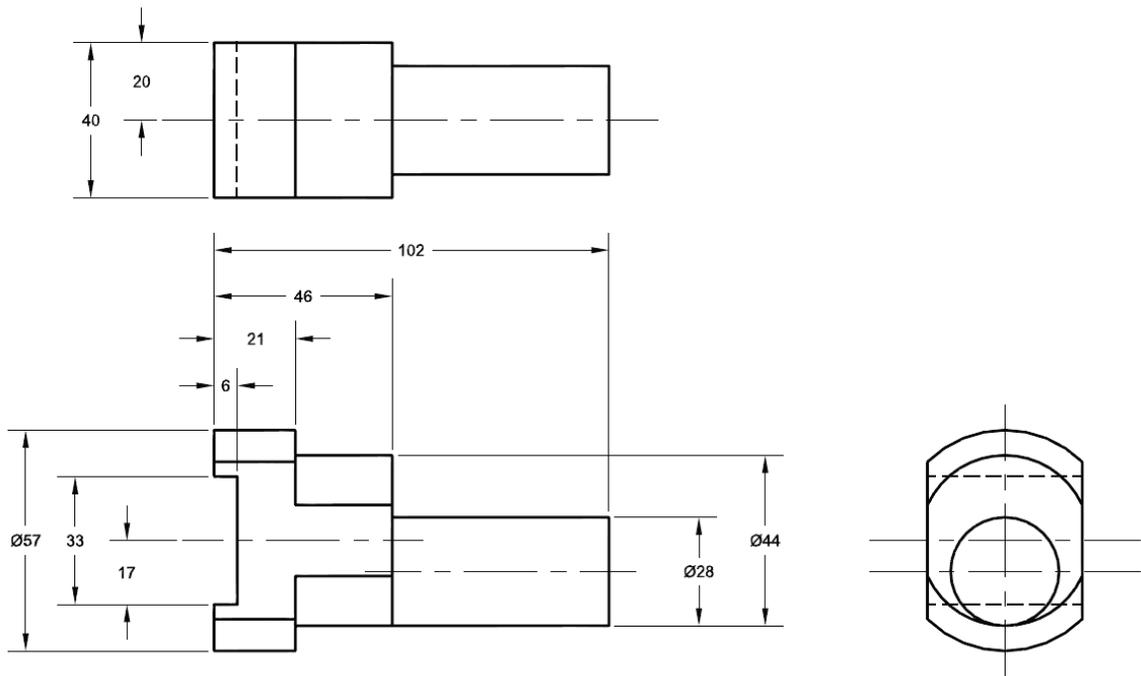
**P2-10)** Use SOLIDWORKS® to create a solid model of the following 1020 Steel part. Calculate the weight of your part. Dimensions given in inches.



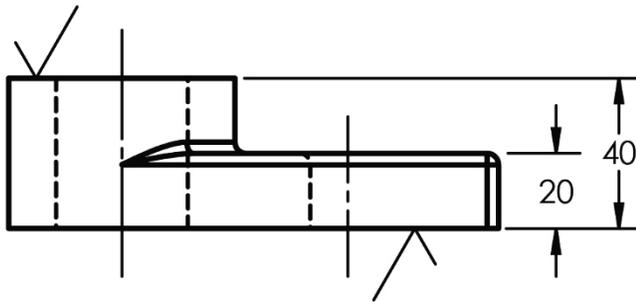
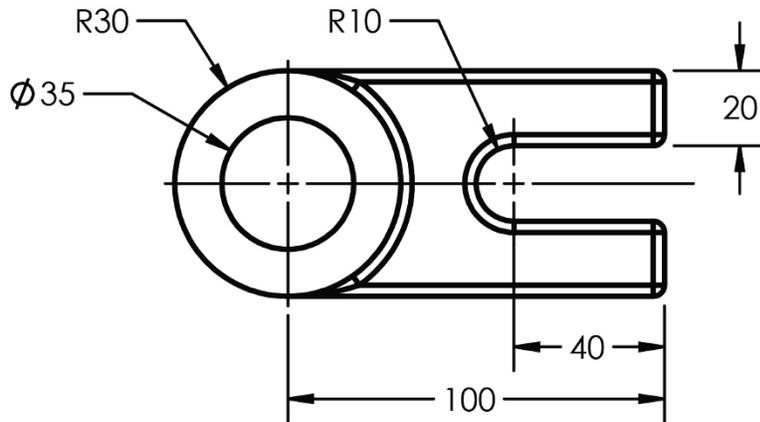
**P2-11)** Use SOLIDWORKS® to create a solid model of the following ABS plastic part. Calculate the weight of your part. Dimensions given in millimeters.



**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.



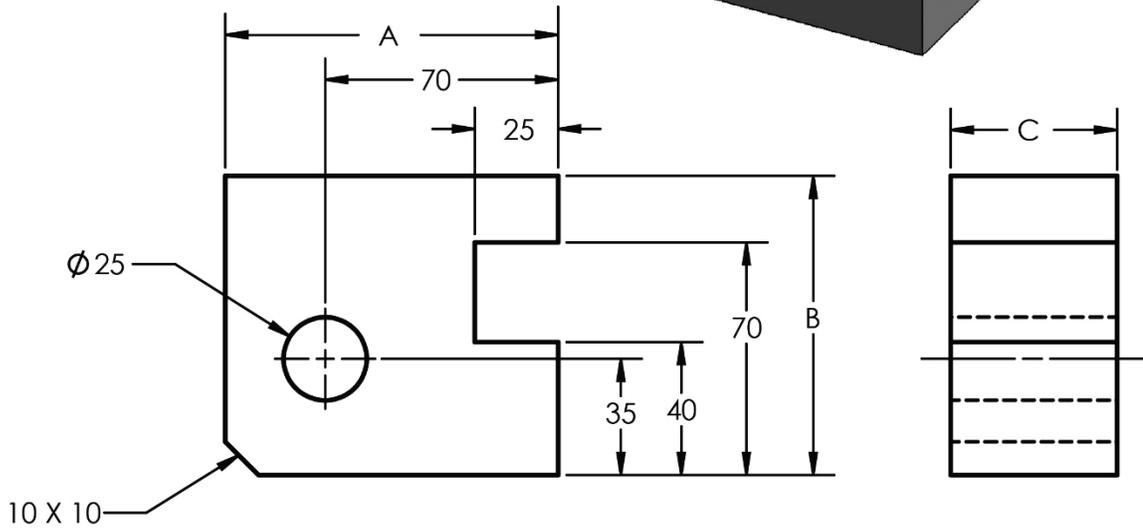
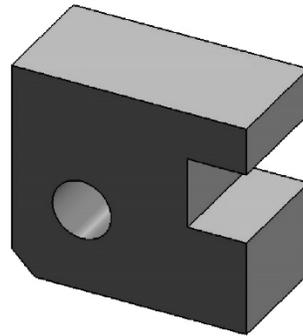
## CONNECTING ROD PROJECT (MODEL) QUIZ PROBLEMS

**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

<u>Group 1</u>	<u>Group 2</u>	<u>Group 3</u>
A = 100	A = 110	A = 90
B = 90	B = 100	B = 95
C = 50	C = 70	C = 40



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

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

