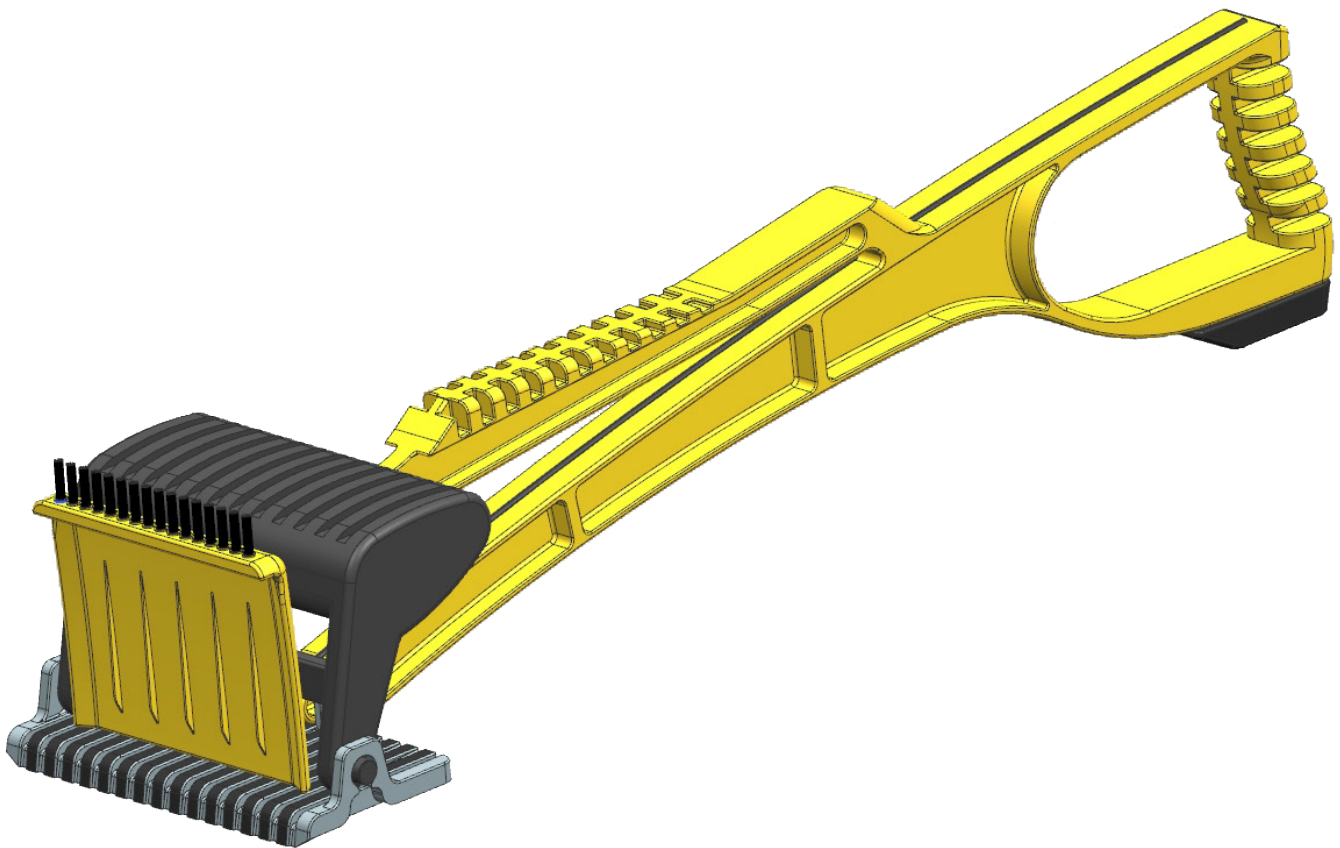


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CHAPTER 2

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	16

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

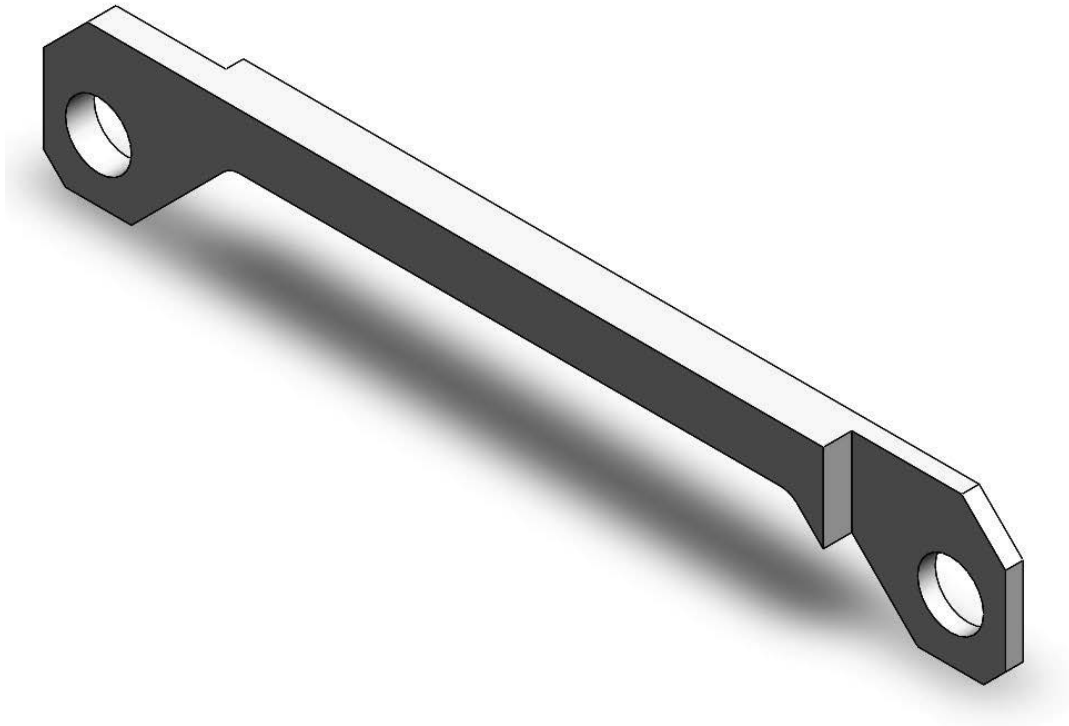


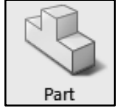



Figure 2.2-1: Connecting rod

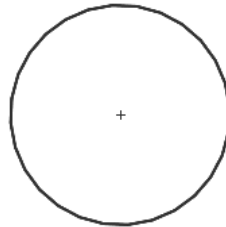
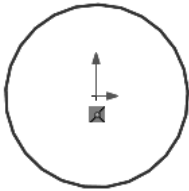
2.3) SETTING UP THE PROJECT

- 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**.
- 3) 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.


Panning

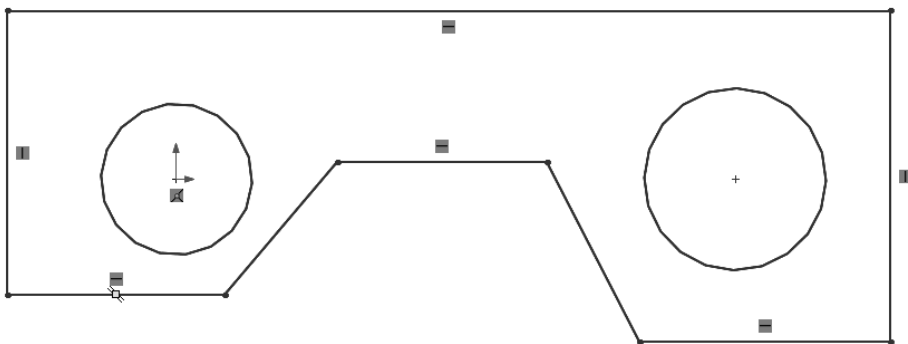
Method 1

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

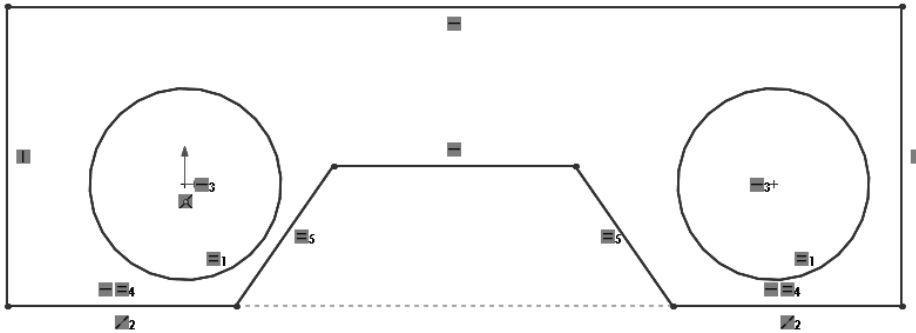
Method 2

- 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.
 - a) Make the two circle diameters **Equal**.
 - b) Make the circle centers **Horizontal**.
 - c) Make the two bottom horizontal lines **Collinear**.
 - d) Make the two bottom horizontal line lengths **Equal**.
 - e) 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.



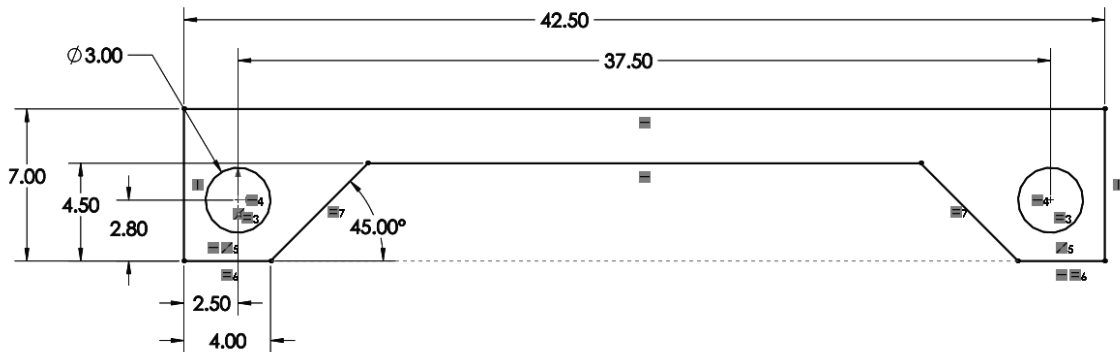
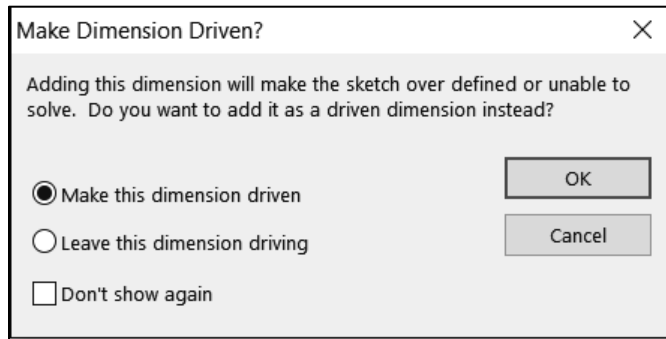
- 5) **View** your **Sketch Relations**. Your geometric relations will show up as symbols inside a green box.

Viewing relations

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




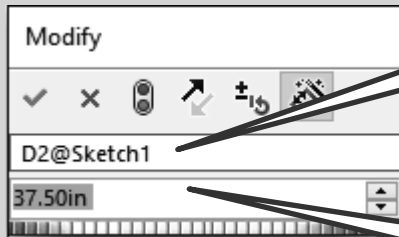
- 6) Add the **Dimensions** shown in the figure shown below. If your drawing exceeds your viewing area, select the **F** key. When dimensioning, it is a good idea to start with the overall dimensions and then work down to the smaller dimensions. Note that if a *Make Dimension Driven?* window appears, 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.



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

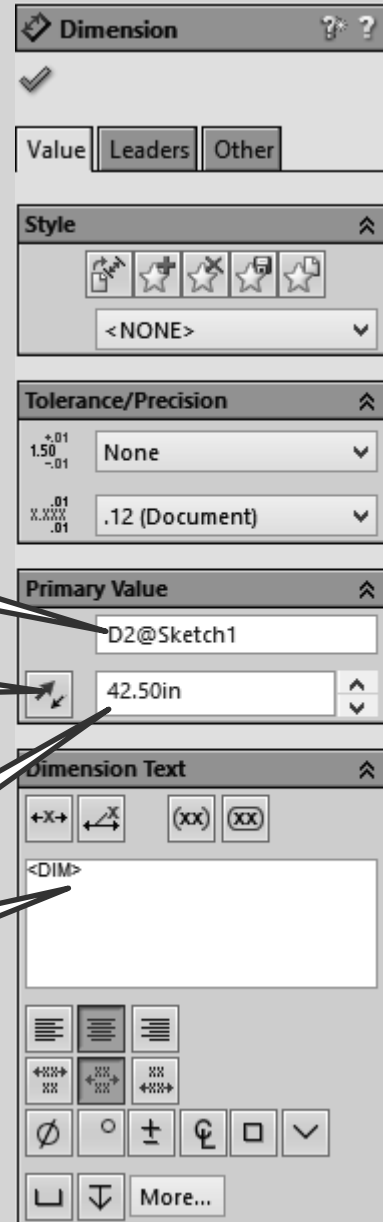
- 1) Select the dimension that you wish to edit.
- 2) Features of the dimension may be changed within the *Dimension* window that appears on the left.

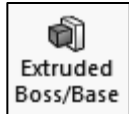
Dimension name

Reverse direction

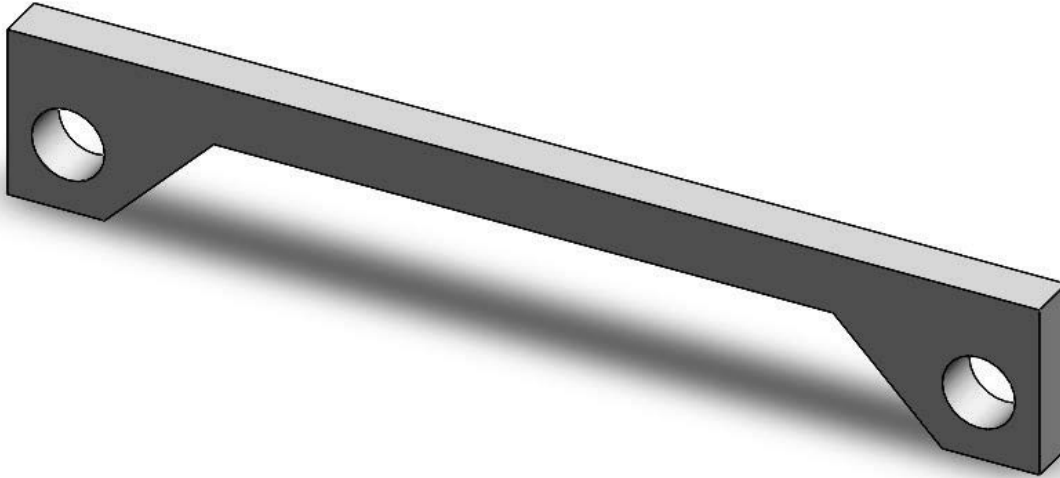
Value

Add symbols and text





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




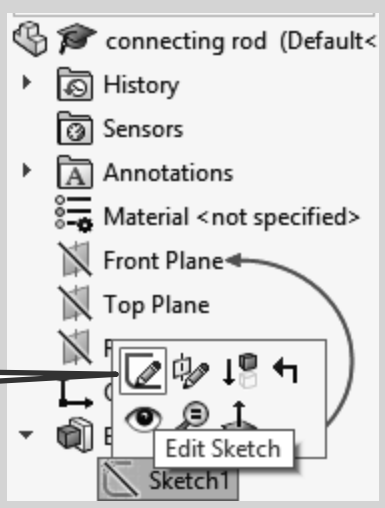
8) Try **zooming in and out** by scrolling your middle mouse wheel. Notice that the mouse location identifies the zooming center.

9) Fit all (**F**).

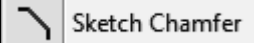
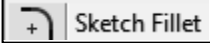
10) **Edit** your sketch.

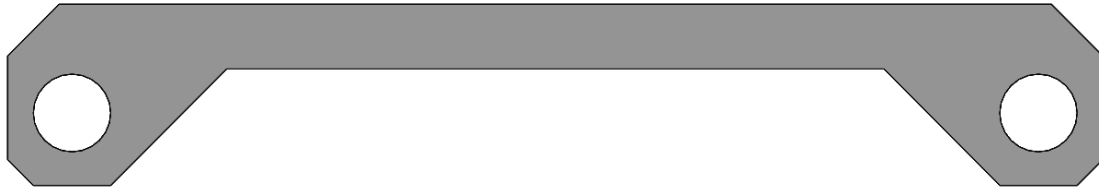
Editing a sketch



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

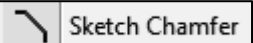
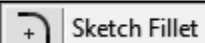
A screenshot of the software interface. On the right is the Feature Manager Design Tree, showing a hierarchy: connecting rod (Default), History, Sensors, Annotations, Material <not specified>, Front Plane, Top Plane, and Sketch1. A callout box labeled "Edit sketch" points to the "Edit Sketch" icon in the Context toolbar, which is located below the Sketch1 feature. The "Edit Sketch" icon is a pencil inside a square.

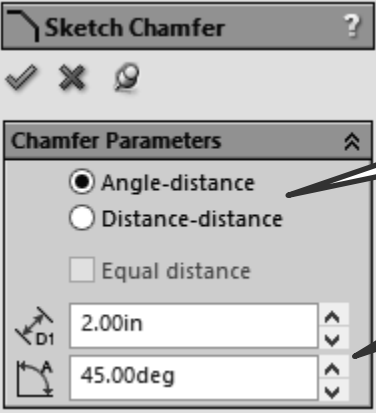
11) View your sketch from the normal plane by selecting **Ctrl + 8**.

- 12) 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. Note that applying the bottom chamfers will delete the *Equal* relation between the two bottom horizontal lines. Reapply the **Equal** relation. See information block on how to create chamfers.



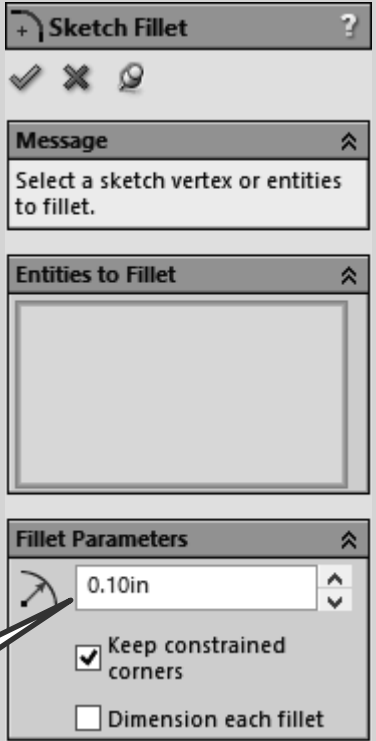
Sketch Chamfers  & **Sketch Fillets**  (located in the *Sketch* tab)

- 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
- D1: 2.00in
- 45.00deg



Sketch Fillet

Message: Select a sketch vertex or entities to fillet.

Entities to Fillet

Fillet Parameters

- 0.10in
- Keep constrained corners
- Dimension each fillet

Chamfer method (points to Angle-distance)

Chamfer dimensions (points to 2.00in and 45.00deg)

Fillet radius (points to 0.10in)

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.



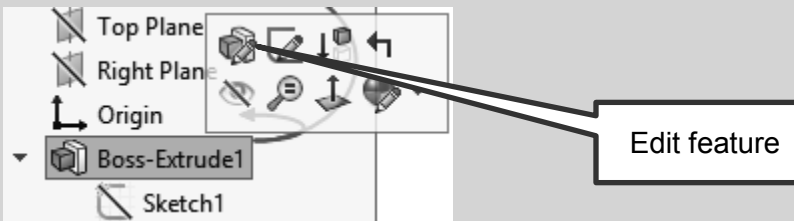
13) **Exit Sketch**

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

15) Change the **Extrude** distance from 2.5 to **2.1 inches**.

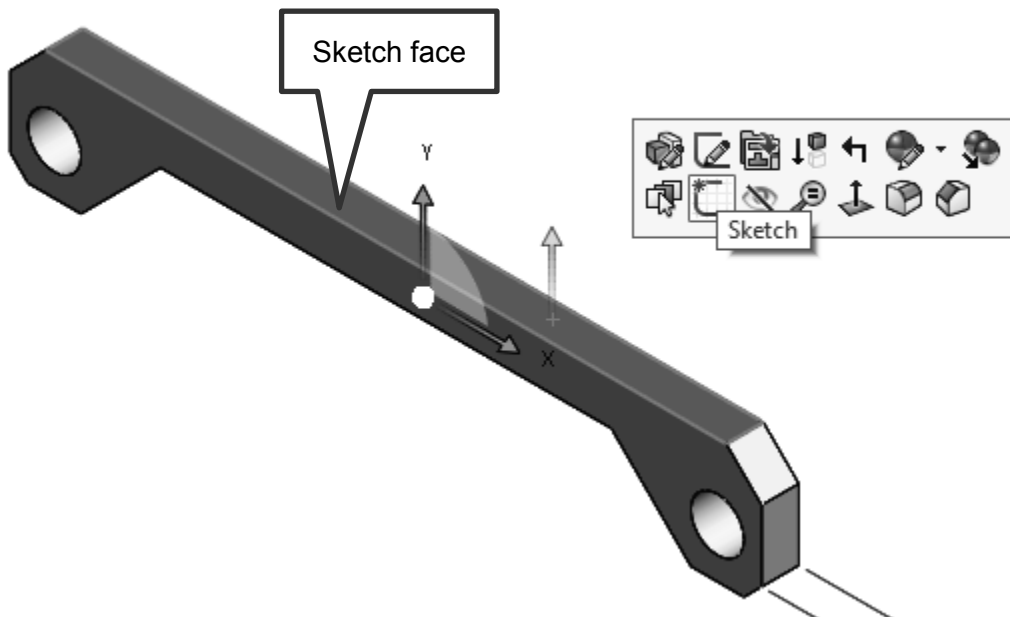
Editing a feature

- 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.



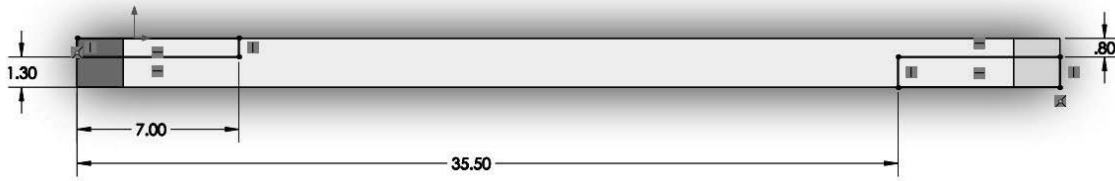
2.5) ADDING FEATURES

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

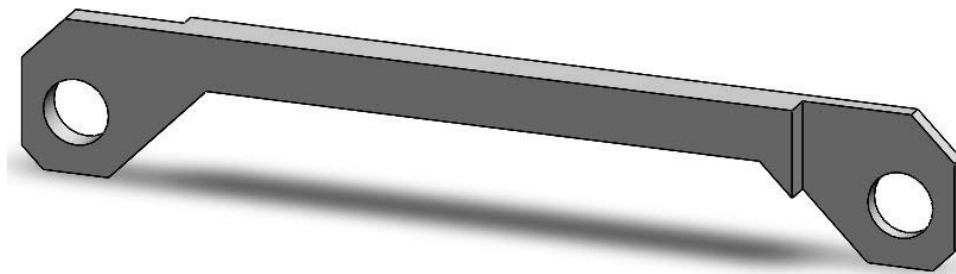


2) View the sketch from the normal direction (**Ctrl + 8**).

- 3) Sketch and dimension the following two **Rectangles** **Corner Rectangle**.



- 4) **Extrude Cut** **Extruded 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.





Rotating your part

Method 1



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

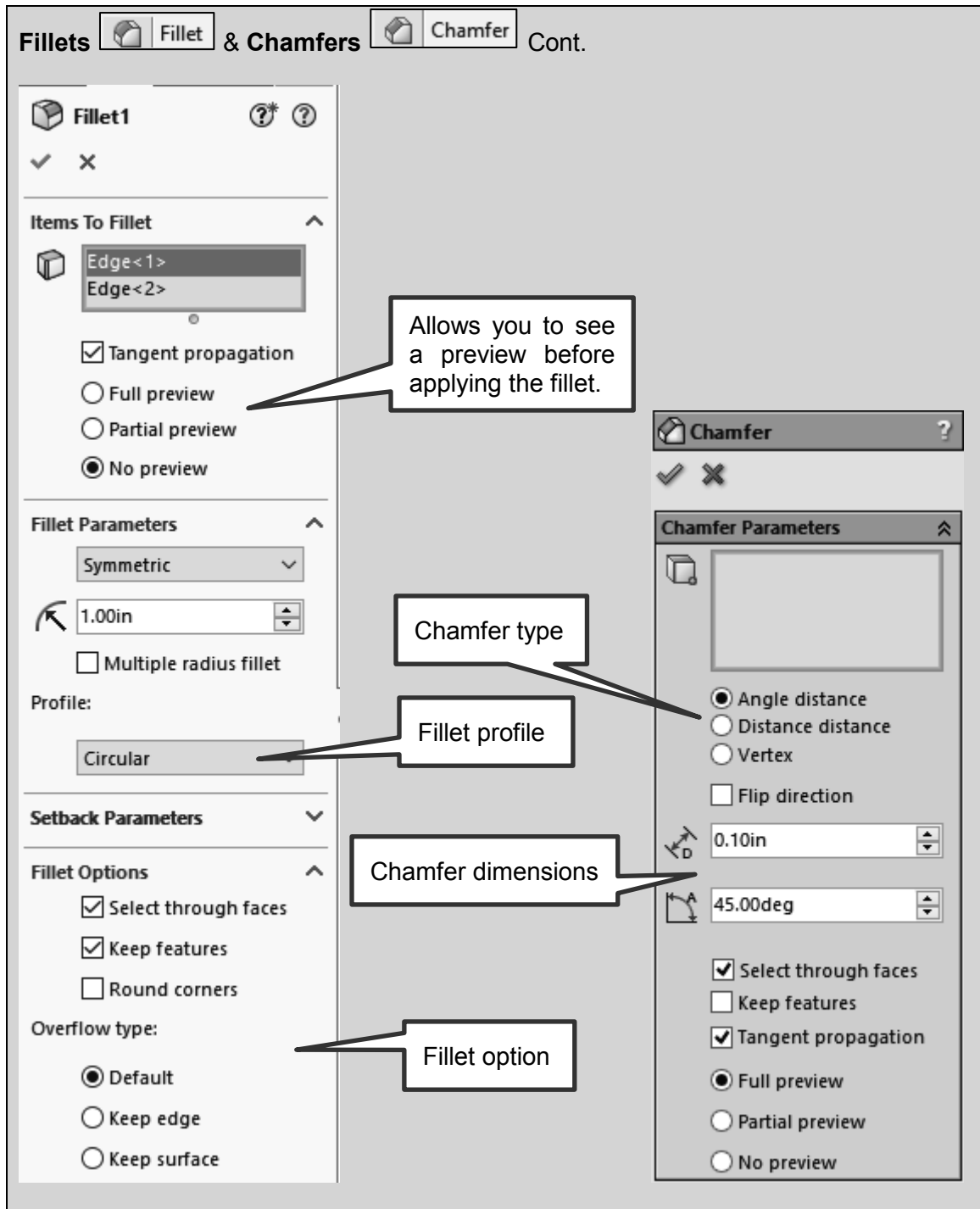
Method 2

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

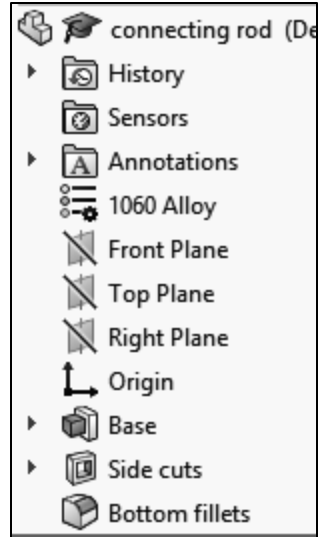
Fillets  **Fillet** & **Chamfers**  **Chamfer** (located in the *Features* tab)

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:**  **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:**  **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.**

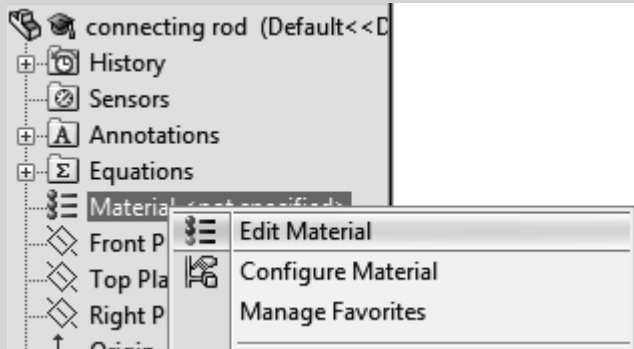


2.6) ADDING MATERIAL

- 1) Apply a material of **Aluminum 1060 Alloy** to your part.

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 Data

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/m² (Pa)

Category: Aluminium Alloys

Name: 1060 Alloy

Default failure criterion: Max von Mises Stress

Description:

Source:

Sustainable:

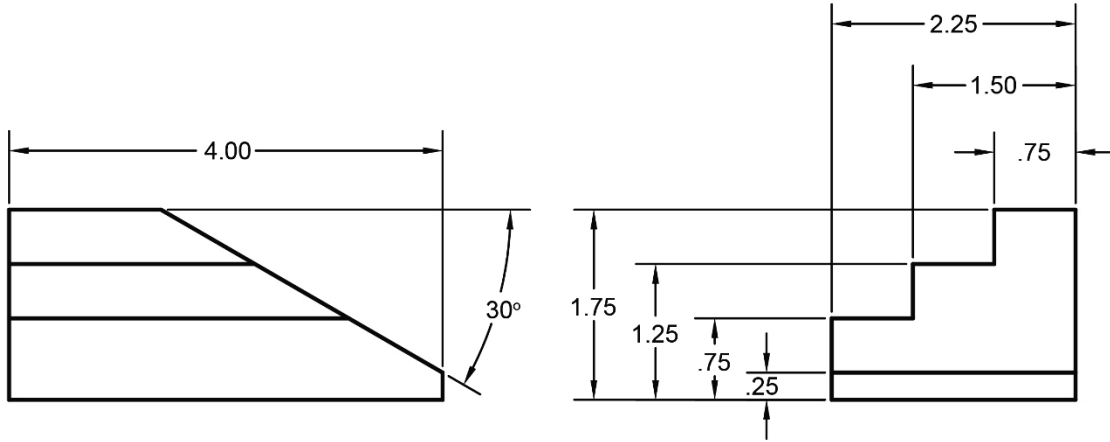
Property	Value	Units
Elastic Modulus	6.9e+010	N/m ²
Poisson's Ratio	0.33	N/A
Shear Modulus	2.7e+010	N/m ²
Mass Density	2700	kg/m ³

- 2) Calculate the weight of your part. Select **Tools – Evaluate - Mass Properties...** In the *Mass Properties* window, note that the weight of your part is 22.22 lb. If your weight is not this value, your model is incorrect. This window also gives other physical properties.

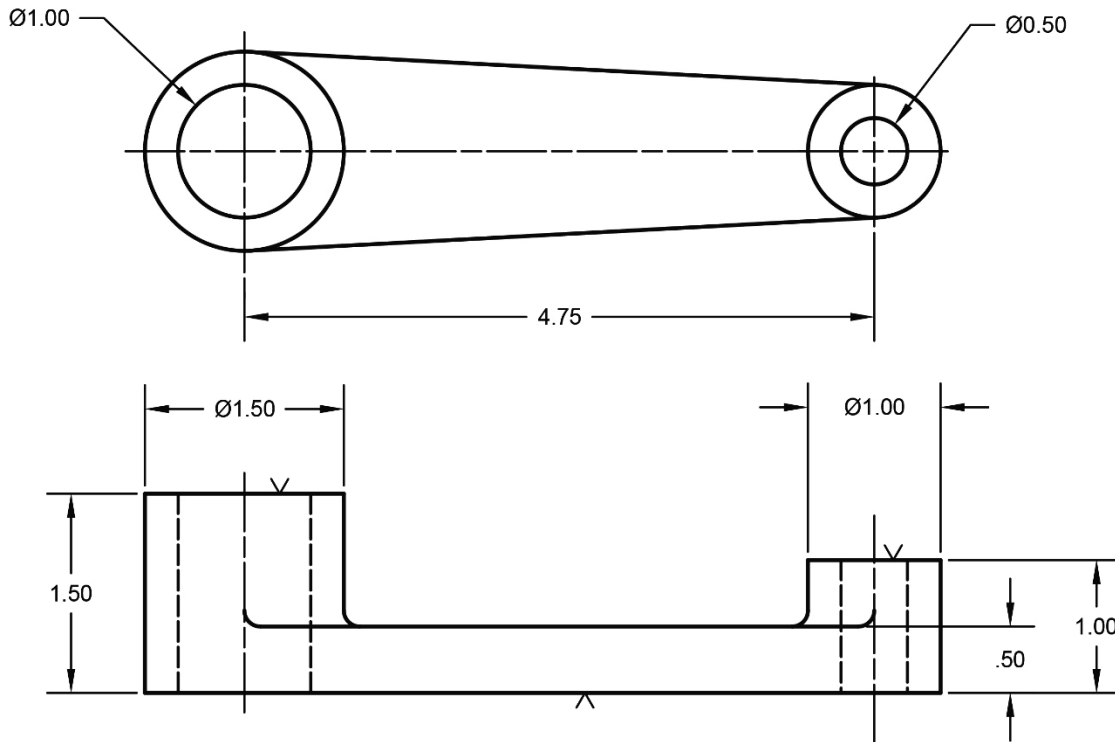
NOTES:

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.

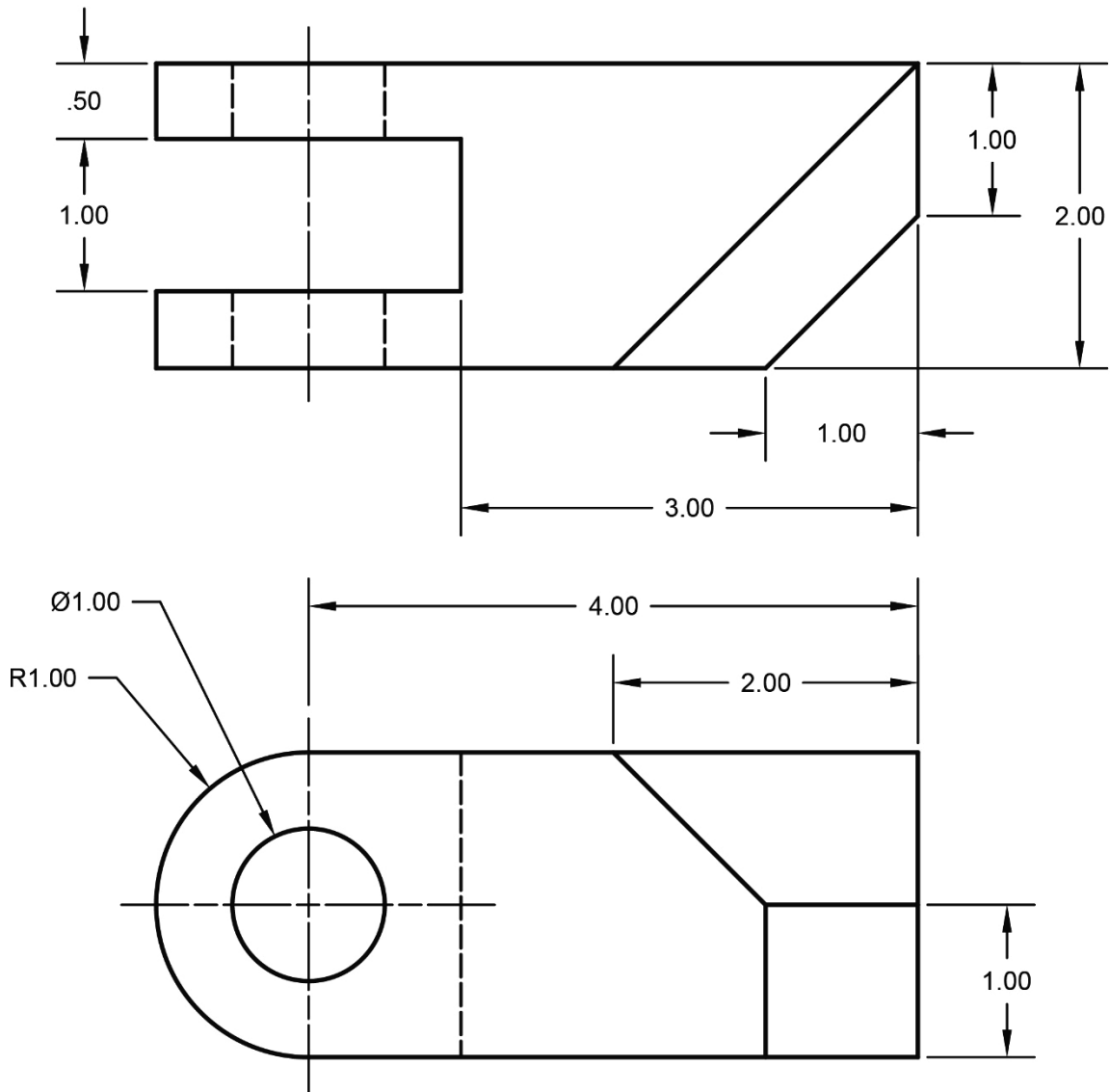


P2-2) Create a solid model of the following Gray Cast Iron part and calculate the weight of the part. Dimensions are given in inches.

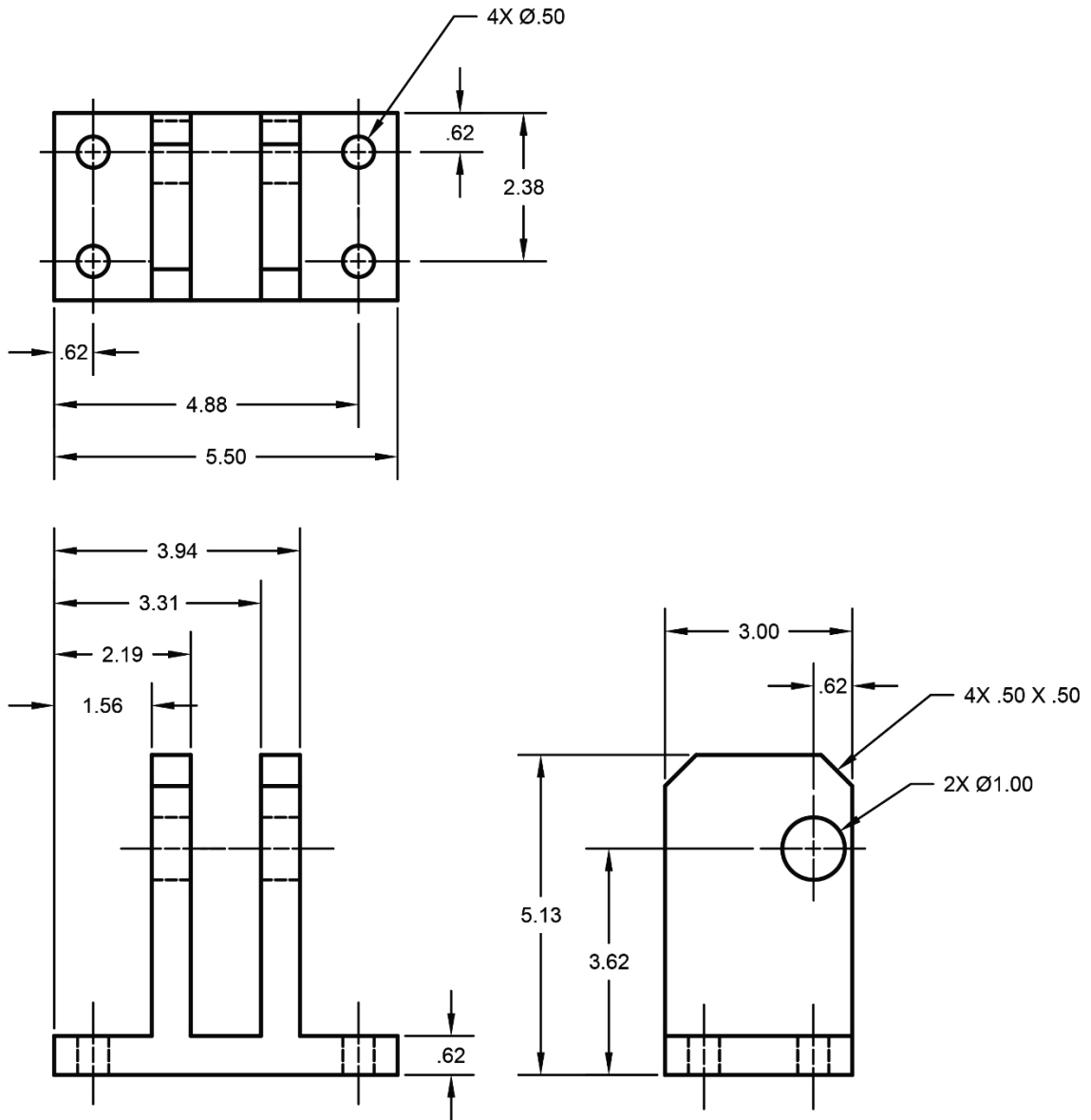


NOTE: ALL FILLETS AND ROUNDS R.12
UNLESS OTHERWISE SPECIFIED

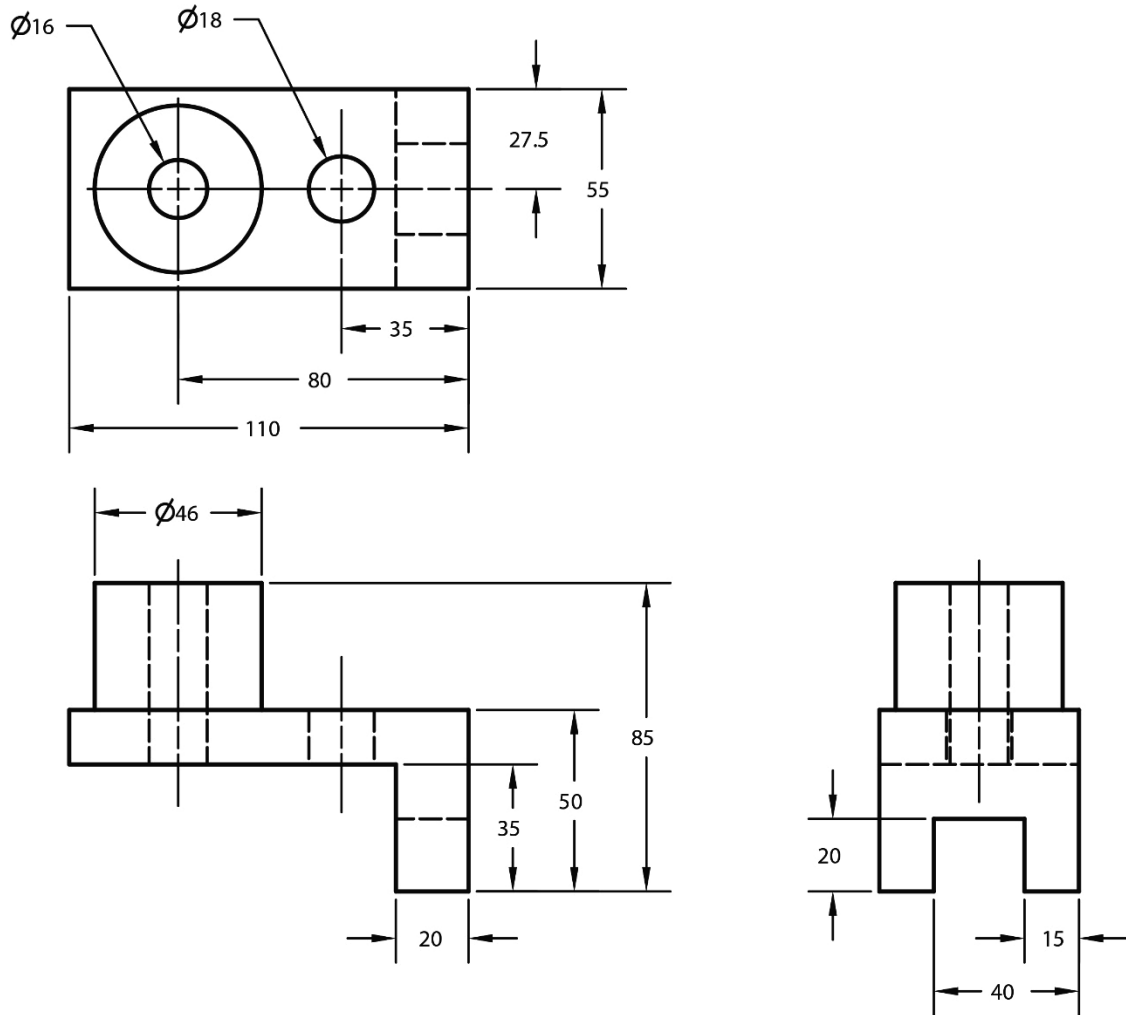
P2-3) Create a solid model of the following 1020 Steel part and calculate the weight of the part. Dimensions are given in inches.



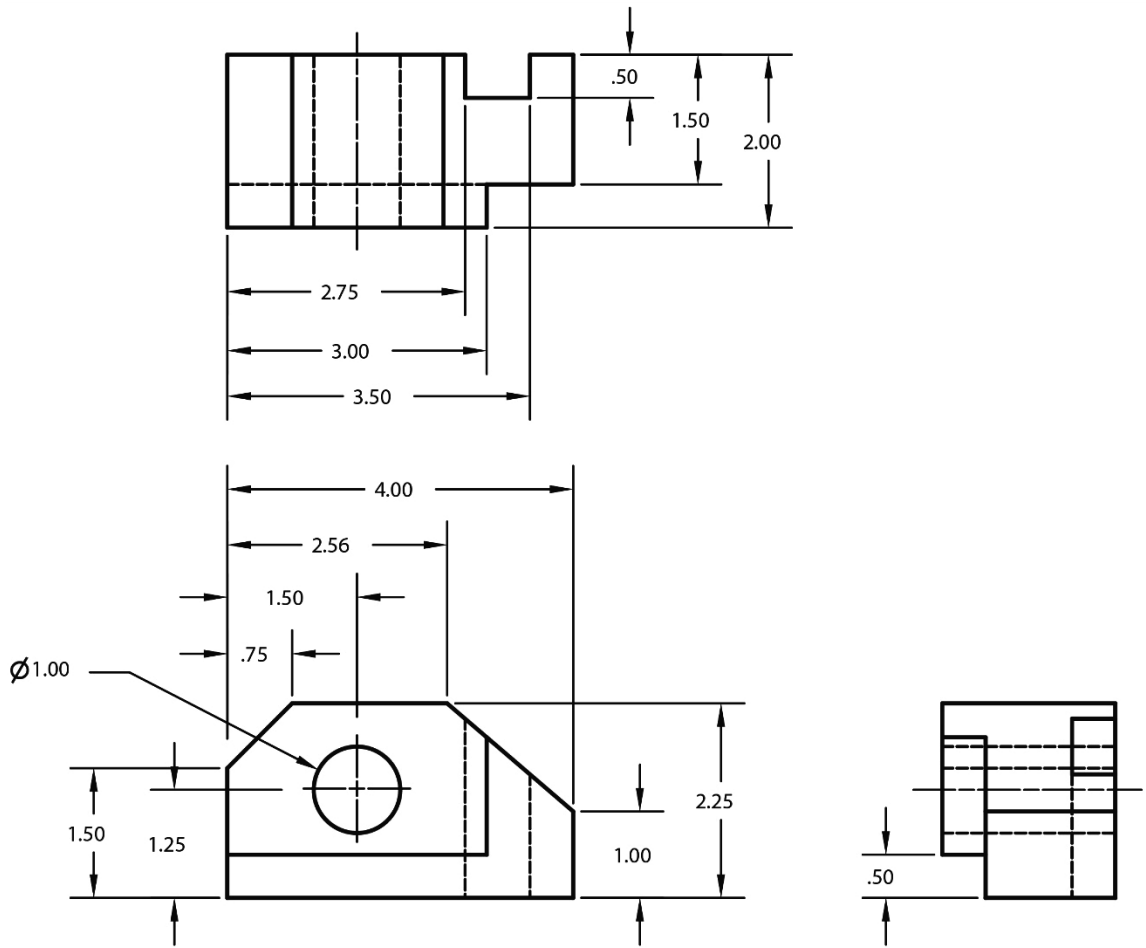
P2-4) Create a solid model of the following ABS plastic part and calculate the weight of the part. Dimensions are given in inches.



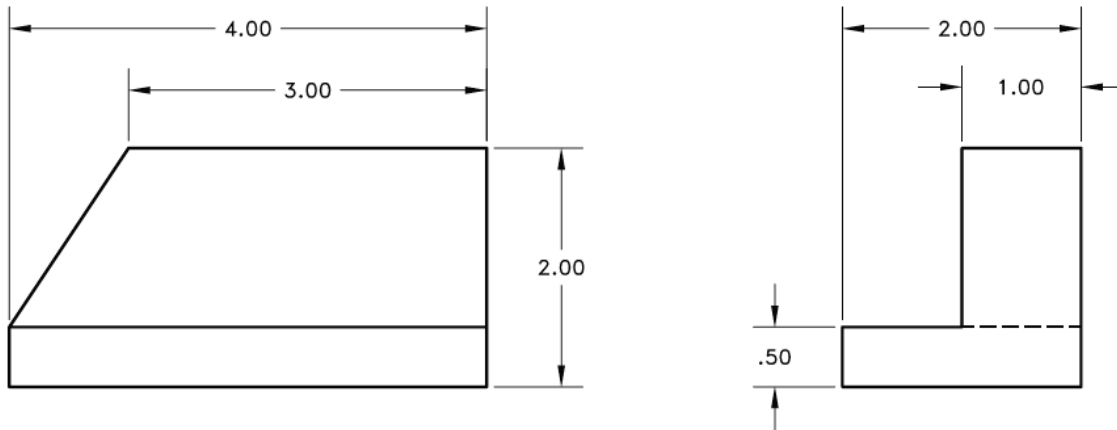
P2-5) Create a solid model of the following Brass part and calculate the mass of the part. Dimensions are given in millimeters.



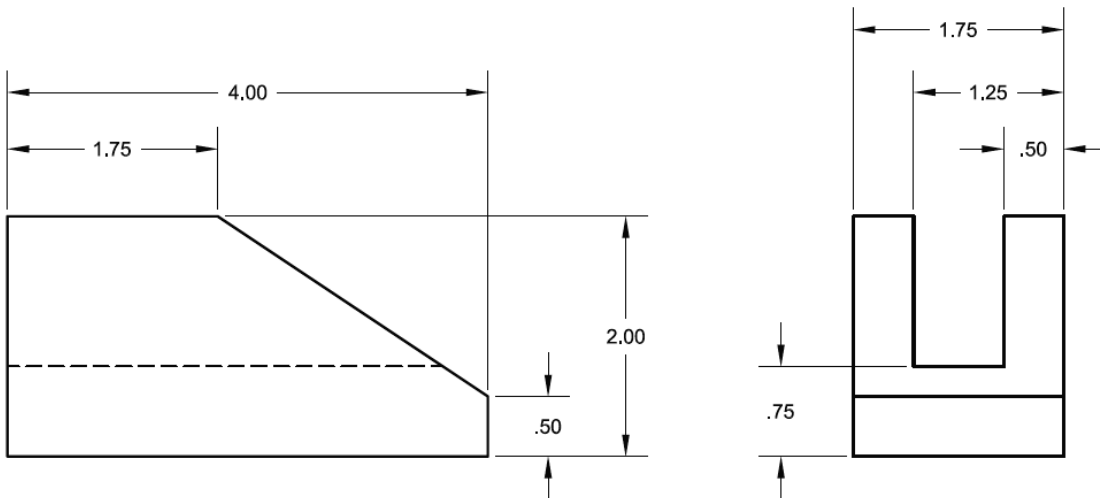
P2-6) Create a solid model of the following Oak part and calculate the weight of the part. Dimensions are given in inches.



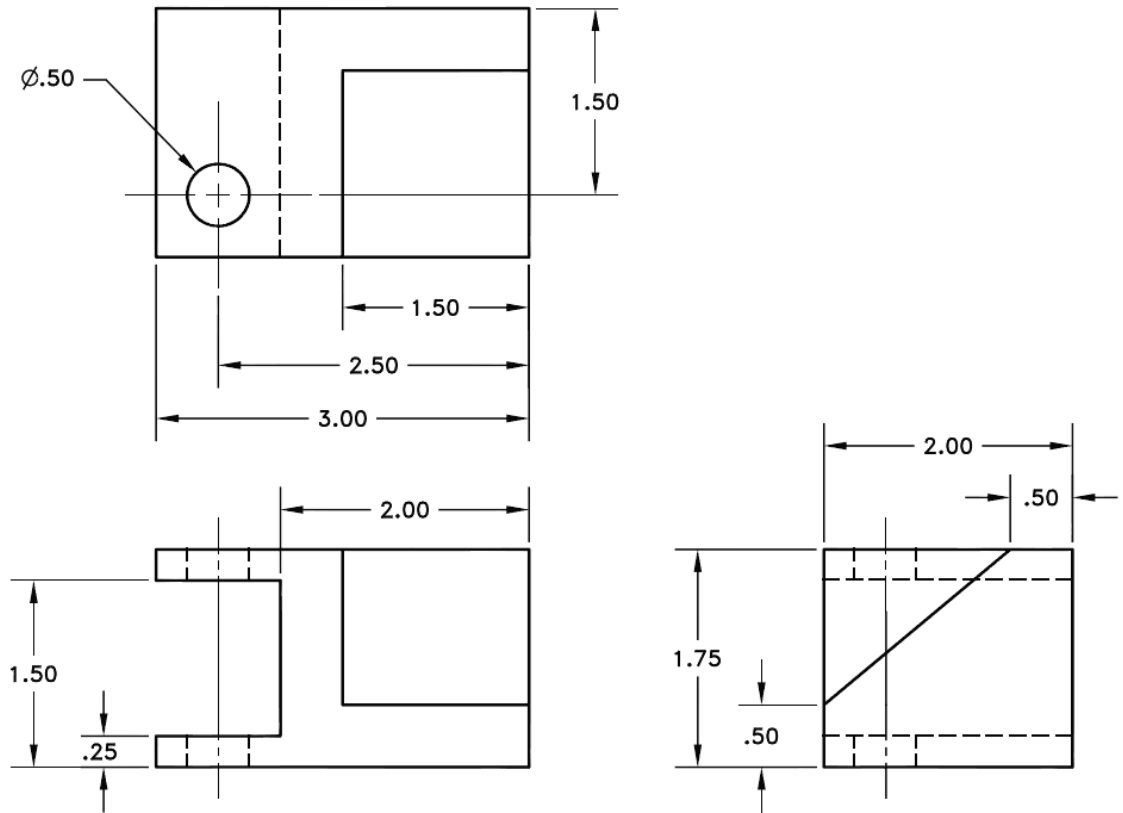
P2-7) Use SOLIDWORKS® to create a solid model of the following 1345 Aluminum part. Calculate the weight of your part. Dimensions are given in inches.



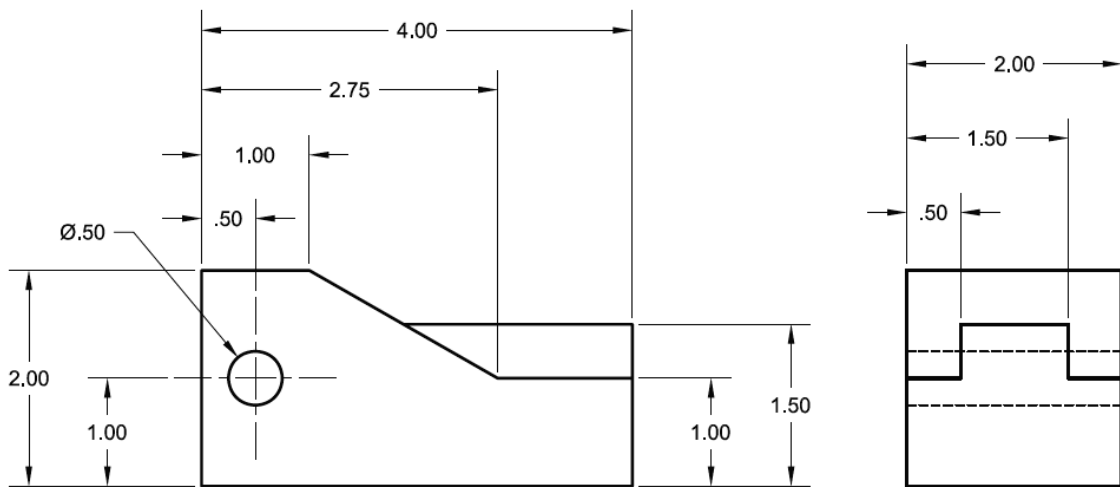
P2-8) Use SOLIDWORKS® to create a solid model of the following 6061 Aluminum part. Calculate the weight of your part. Dimensions given in inches.



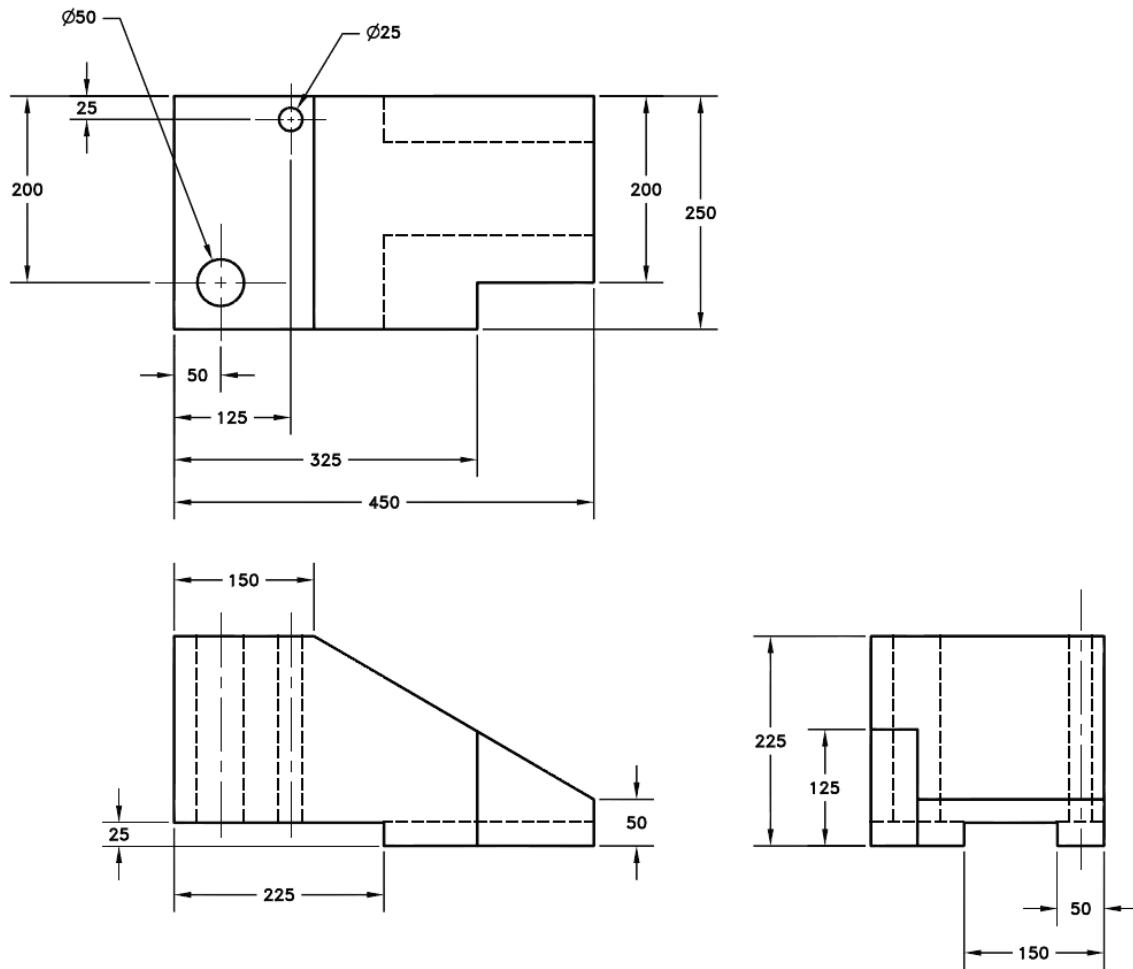
P2-9) Use SOLIDWORKS® to create a solid model of the following 1020 Steel part. Calculate the weight of your part. Dimensions given in inches.



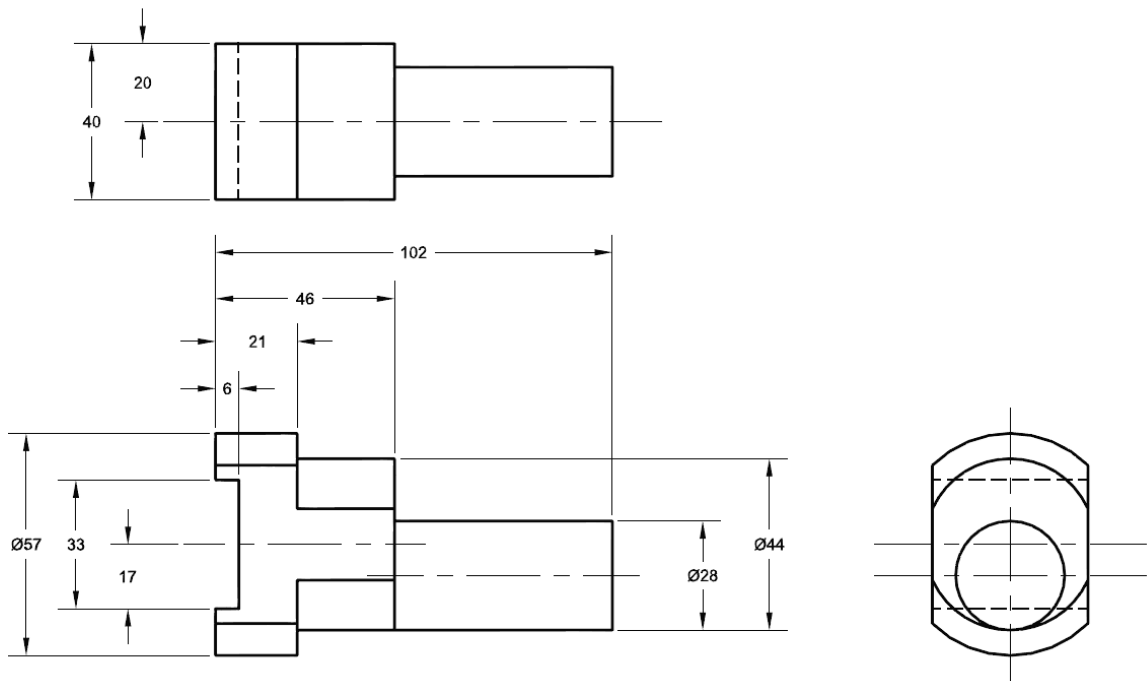
P2-10) Use SOLIDWORKS® to create a solid model of the following 1020 Steel part. Calculate the weight of your part. Dimensions given in inches.



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.

