Project Based
SOLIDWORKS® 2020
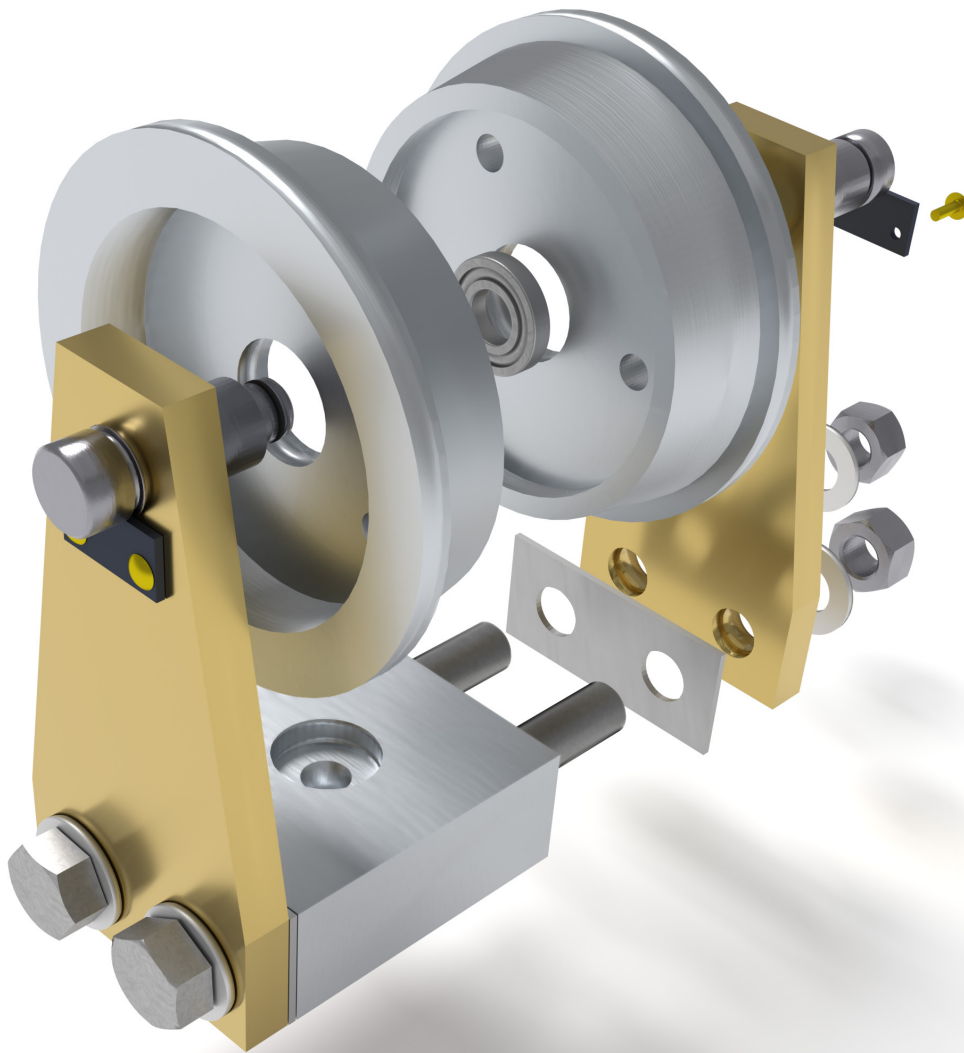
Kirstie Plantenberg, CSWA

Visit the following websites to learn more about this book:

CHAPTER 2

CONNECTING ROD PROJECT
Model

CHAPTER OUTLINE

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

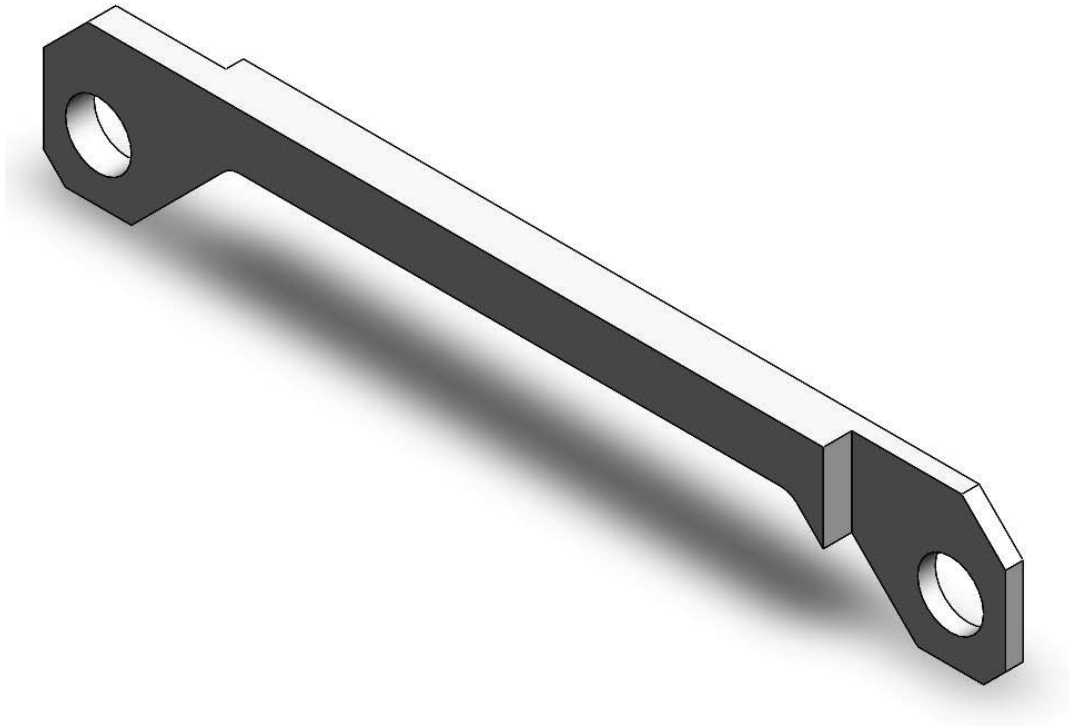- Chamfer
- Fillets
- Editing a feature

Material and properties

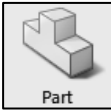- Applying material
- Mass properties

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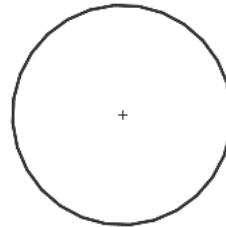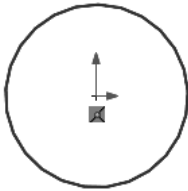
3) **Pan** your drawing area to center the circles. (See the informational block on **Panning**.)

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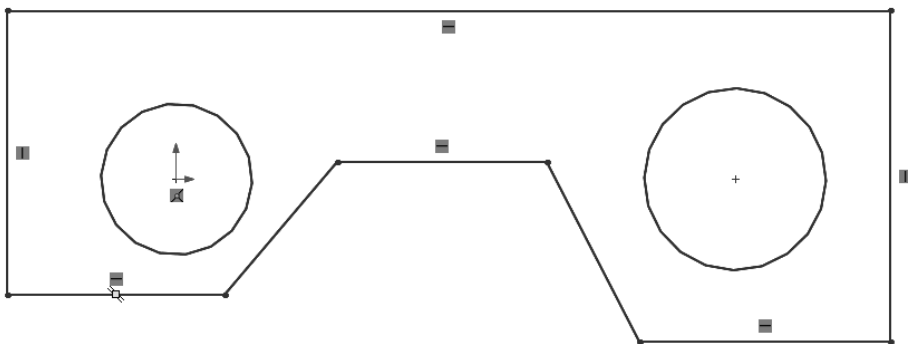
5) Add the following sketch relations. Don't worry if your drawing goes wonky. Just click and drag the elements into position. (See the informational block on ***Applying Sketch Relations***.)
   a) If any of the **horizontal** or **vertical** lines are not perfectly horizontal or vertical, add those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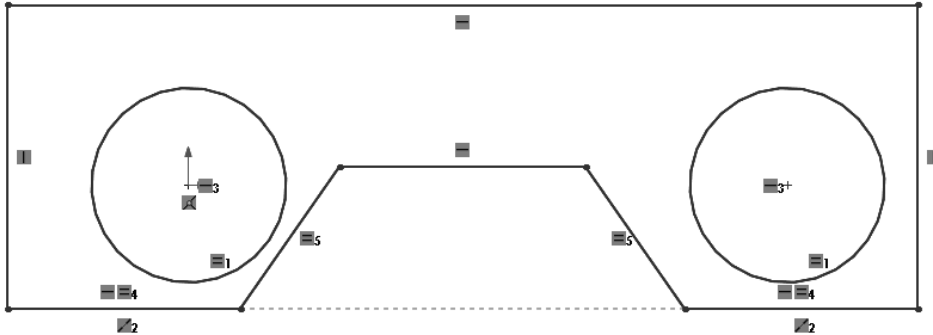
Applying sketch relations

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.
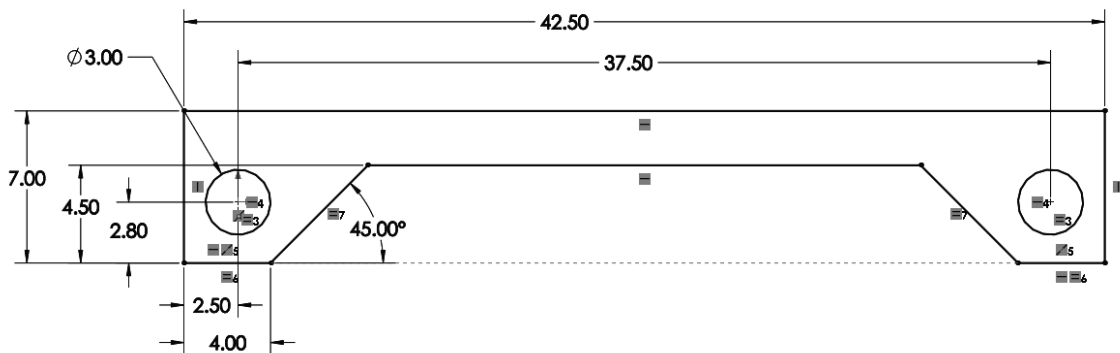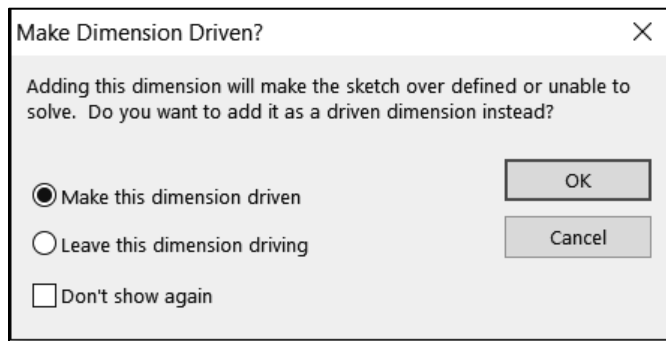4) In the *Properties* window, select the relation that you wish to apply.

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**.

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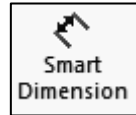
7) Add the **Dimensions** [Smart Dimension] shown in the figure shown below. If your drawing exceeds your viewing area, select the **F** key to fit all. When dimensioning, it is a good idea to start with the overall dimensions and then work down to the smaller dimensions. Note that if a *Make Dimension 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: 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*.)

> **Make Dimension Driven?** ✕
>
> Adding this dimension will make the sketch over defined or unable to solve. Do you want to add it as a driven dimension instead?
>
> ⦿ Make this dimension driven
> ◯ Leave this dimension driving
> ☐ Don't show again
>
> [OK]
> [Cancel]

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.
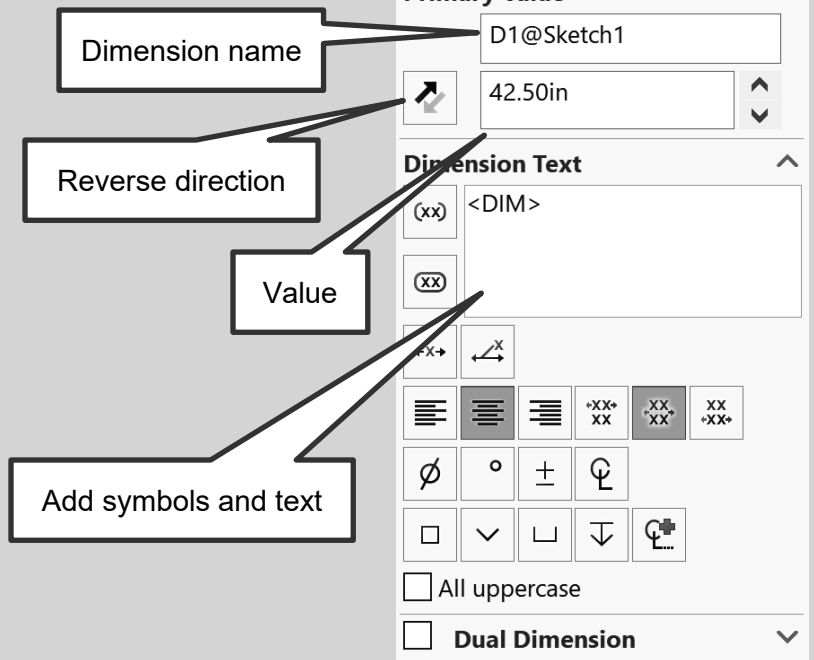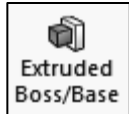
Editing a dimension

Method 1

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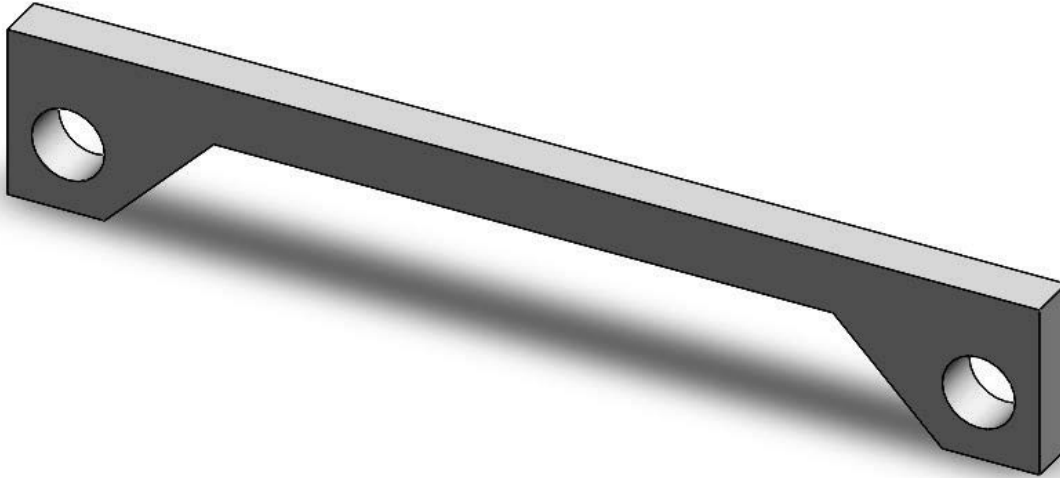2) Within the *Modify* window, fill in the correct dimension value and then select the green check mark [✓].

Modify

Dimension name

D2@Sketch1

37.50in

Value
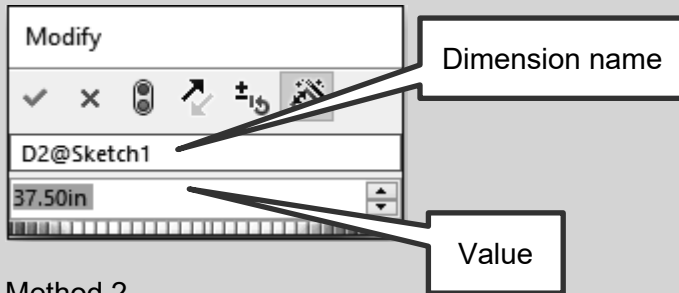
Method 2

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

◇ **Dimension**

✓

Value | Leaders | Other

Style ⌃

<NONE>

Tolerance/Precision ⌃

None

.12 (Document)

Primary Value ⌃

D1@Sketch1

42.50in

Dimension Text ⌃

(xx)   <DIM>

(XX)

⌖X→   ⌖X

∅   °   ±   ℄

☐   ⌄   ⌴   ⟱   ℄

☐ All uppercase

☐ **Dual Dimension** ⌄

9) **Extrude** ![Extruded Boss/Base] your sketch to a distance of **2.50** inches.
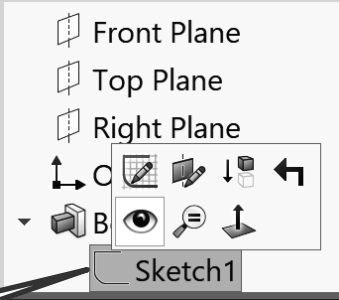
10) Try **zooming in and out** by scrolling your middle mouse wheel. Notice that the mouse location identifies the zooming center.

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

Edit sketch

Sketch1


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

14) Add two **2 x 45º Chamfers** ⬉ Sketch Chamfer to the top outside corners and two **1 x 45º Chamfers** to the bottom 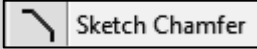outside c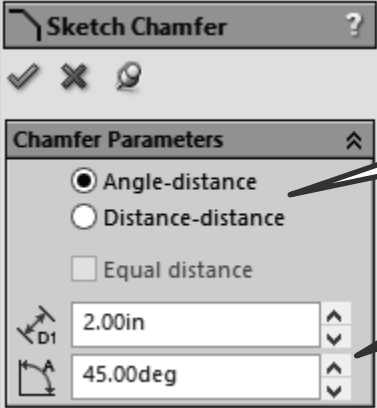orners. The *Chamfer* command is located under the *Fillet* ⬉+ Sketch Fillet command. When applying the 1 x 45º, select **Yes** in the warning window. Note that this will delete the *Equal* relation between the two bottom horizontal lines. Reapply the **Equal** relation after you have applied the chamfer. (See the information block on **Sketch Chamfers & Sketch Fillets**.) After you reapply the Equal relation, you may be asked to **Rebuild and Save**.
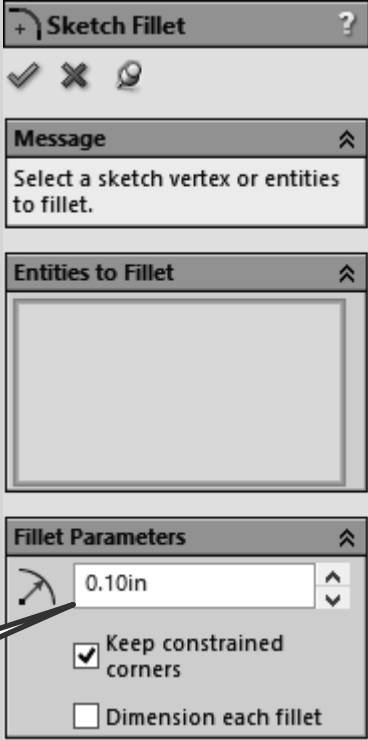
SOLIDWORKS ✕

⚠ At least one sketch constraint is about to be lost. Chamfer anyway?

Yes   No
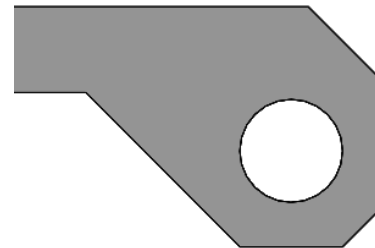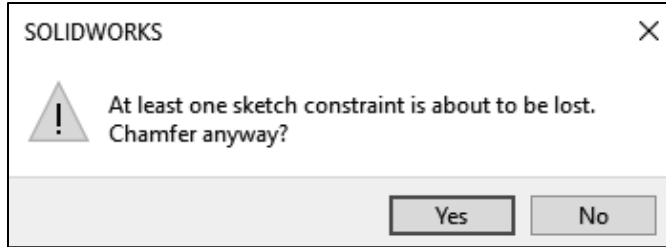
Sketch Chamfers ⬉ Sketch Chamfer & **Sketch Fillets** ⬉+ Sketch Fillet (located in the *Sketch* tab)

1) 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.

2) Fillet: ⬉+ Sketch Fillet A *Fillet* is a rounded corner. It is defined by selecting a vertex or two edges and then specifying a radius.

Sketch Fillet ?
✓ ✗ ⦿

Message ⌃
Select a sketch vertex or entities to fillet.

Entities to Fillet ⌃

Sketch Chamfer ?
✓ ✗ ⦿

Chamfer Parameters ⌃
◉ Angle-distance
○ Distance-distance
☐ Equal distance
D1 2.00in
45.00deg

Chamfer method

Chamfer dimensions

Fillet radius

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.

15) **Exit Sketch** ⌐↵ 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*.)
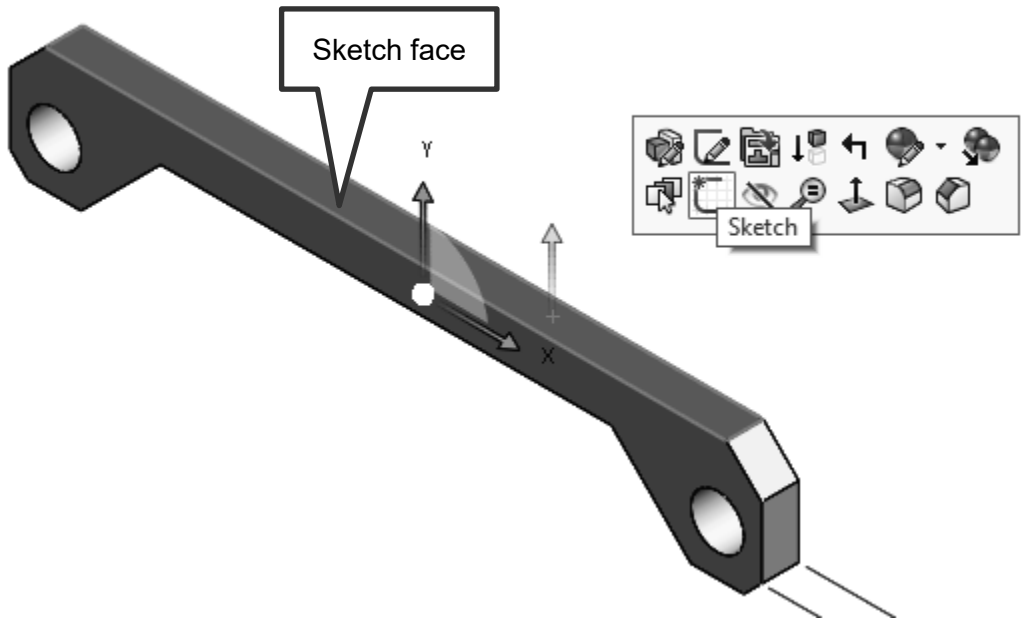

Editing a feature

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.
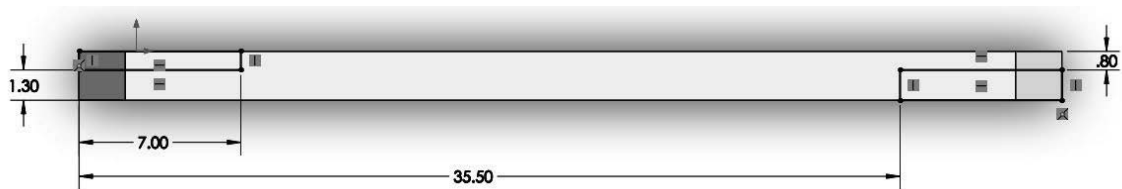

Edit feature

2.5) ADDING FEATURES

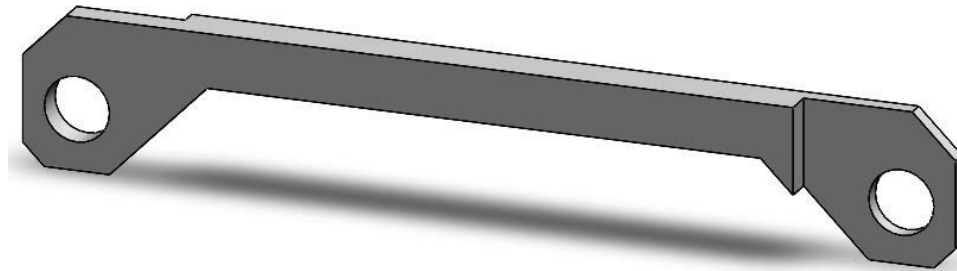1) **Sketch** on the top face of your part.

Sketch face

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*.)

1.30

.80

7.00

35.50

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. (See the informational blocks on *Rotating your part* and *Fillets & Chamfers*.)

Radius: 1in

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.

Fillets [Fillet] **& Chamfers** [Chamfer] Cont.

Fillet

Manual | FilletXpert

Fillet Type

> Fillet type

Items To Fillet

Edge<1>
Edge<2>

☑ Show selection toolbar
☑ Tangent propagation
○ Full preview
○ Partial preview
◉ No preview

> Allows you to see a preview before applying the fillet.

Fillet Parameters

Symmetric

1.00in

☐ Multi Radius Fillet

Profile:

Circular

> Fillet profile

Setback Parameters

☐ **Partial Edge Parameters**

Fillet Options

☑ Select through faces
☑ Keep features
☐ Round corners

Overflow type:

◉ Default
○ Keep edge
○ Keep surface

> Fillet option

Chamfer

Chamfer Type

> Chamfer type

Items To Chamfer

☑ Tangent propagation
○ Full preview
○ Partial preview
◉ No preview

Chamfer Parameters

☐ Flip direction

0.39370079in

45.00deg

> Chamfer dimensions

Chamfer Options

☑ Select through faces
☑ Keep features

2 - 13

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

7) **Save**.

CONNECTING ROD
▸ 📦 Solid Bodies(1)
▸ 🅰 Annotations
▸ Σ Equations
🔧 1060 Alloy
📄 Front Plane
📄 Top Plane
📄 Right Plane
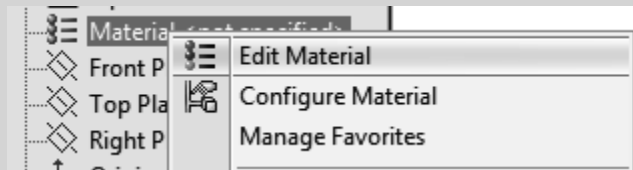↳ Origin
▸ 🪑 Base
▸ 📦 Side Cuts
📦 Bottom Fillets
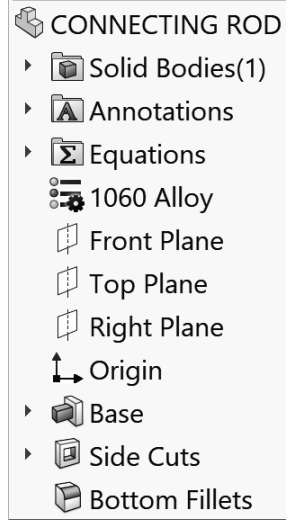
2.6) ADDING MATERIAL

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
Front P   §≡ Edit Material
Top Pla   ⊠ Configure Material
Right P   Manage Favorites

Material [×]

SolidWorks DIN Materials
SolidWorks Materials
  Steel
  Iron
  Aluminium Alloys
    1060 Alloy
    1060-H12
    1060-H12 Rod (SS)
    1060-H14
    1060-H16
    1060-H18
    1060-H18 Rod
    1060-O (SS)
    1100-H12 Rod
    1100-H16 Rod (
    1100-H26
    1100-O Ro
    1345 Alloy
    1350 Alloy
    201.0-T43 Insulated Mold Casting (SS)
    201.0-T6 Insulated Mold Casting (SS)
    201.0-T7 Insulated Mold Casting (SS)

Properties | Tables & Curves | Appearance | CrossHatch | Custom | Application Dat

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^2 (Pa)
Category: Aluminium Alloys
Name: 1060 Alloy
Default failure   Max von Mises Stress
criterion:
Description:
Source:
Sustainab

Units

Select material

Material properties

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. 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.
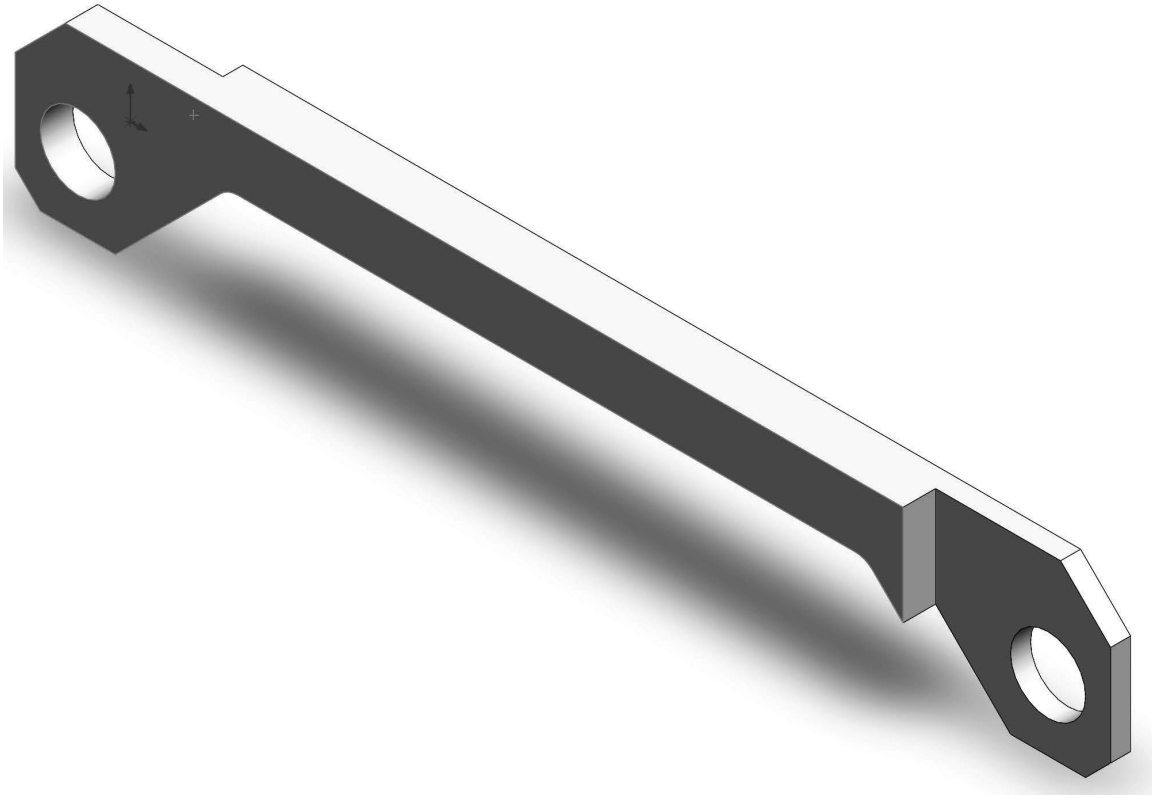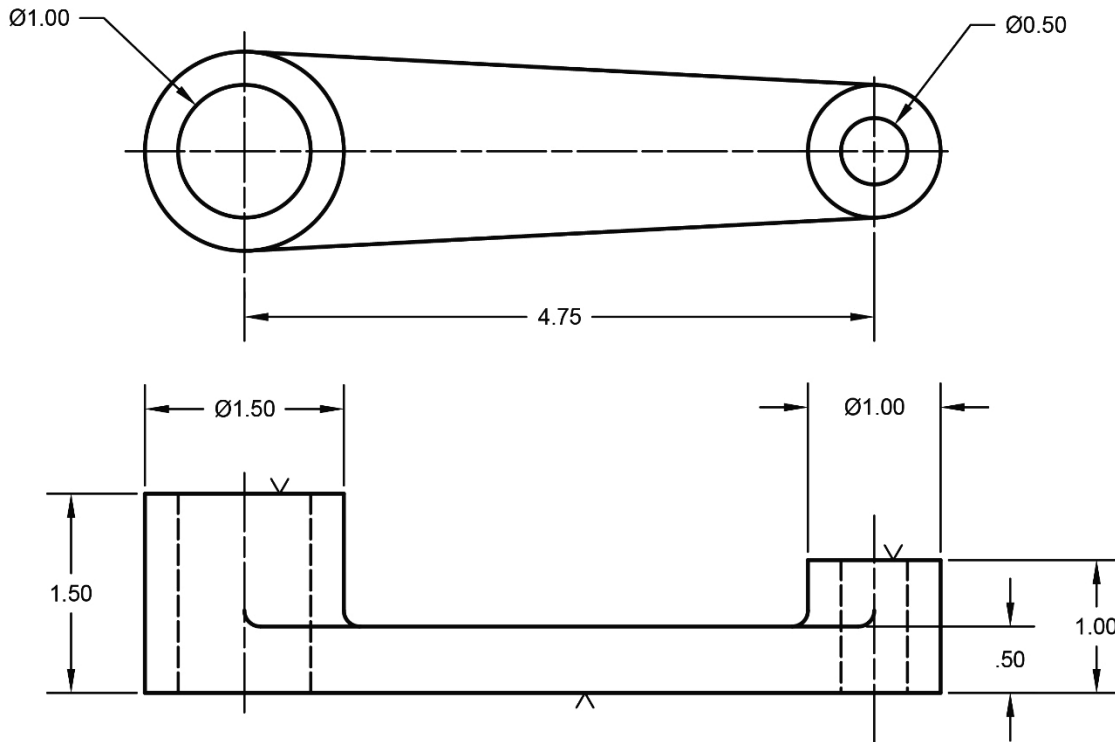
NOTES:

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.
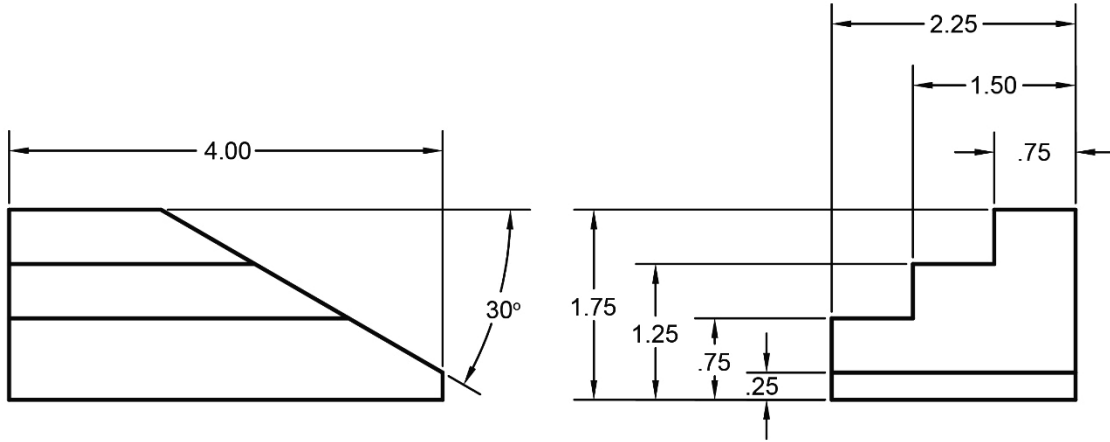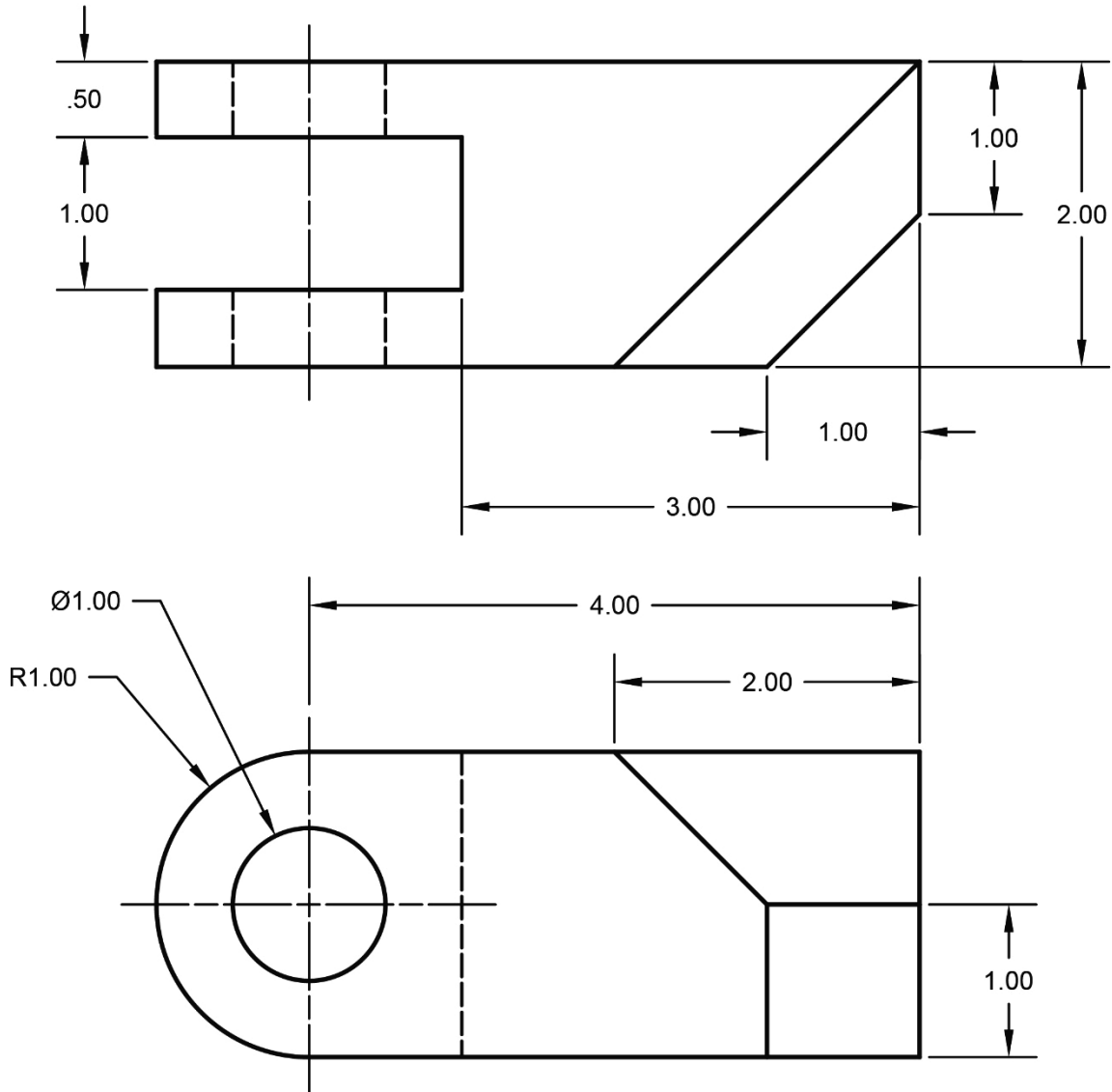

P2-2) Create a solid model of the following Gray Cast Iron part and calculate the weight of the part. Dimensions are given in inches.
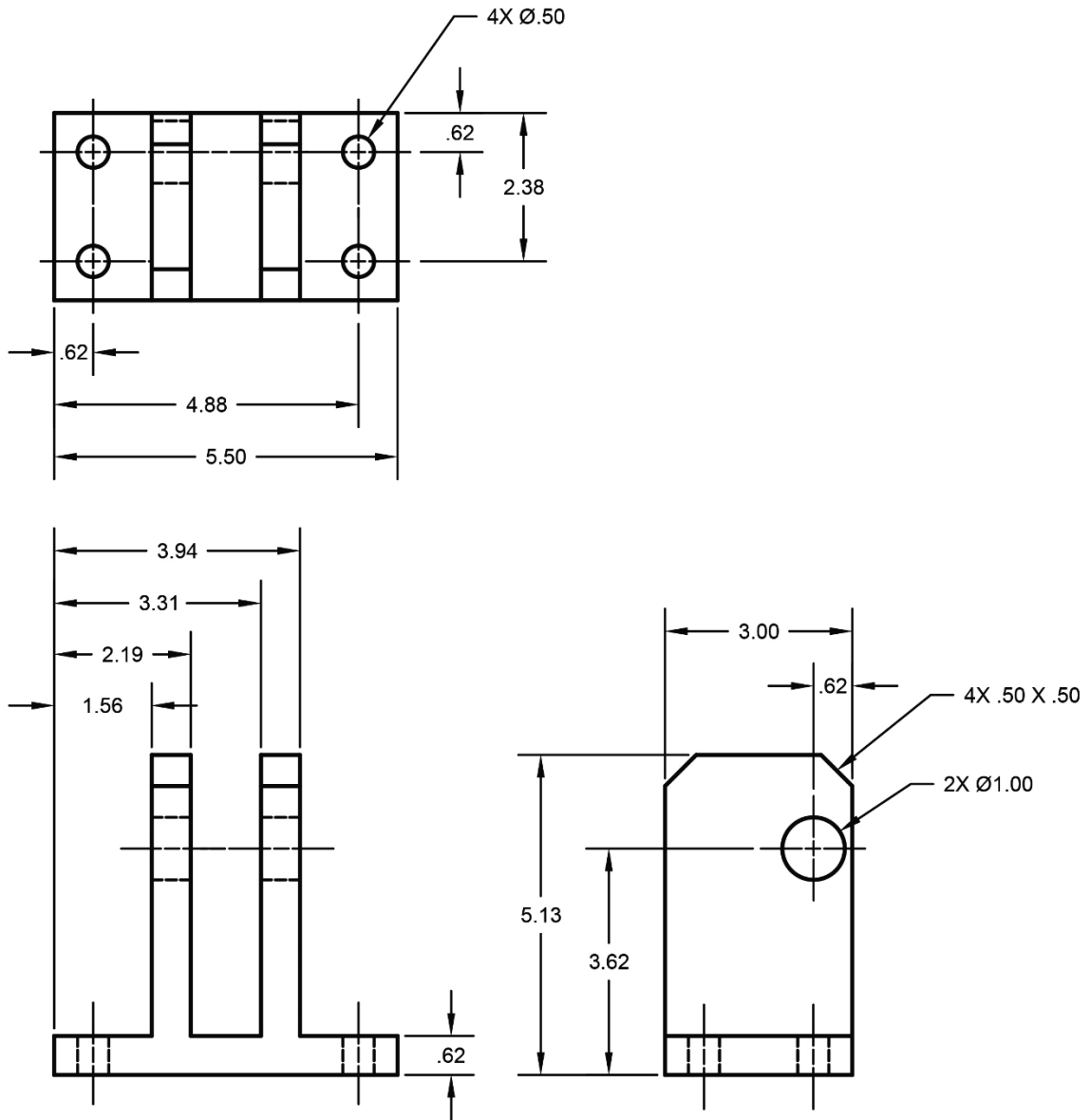

NOTE: ALL FILLETS AND ROUNDS R.12
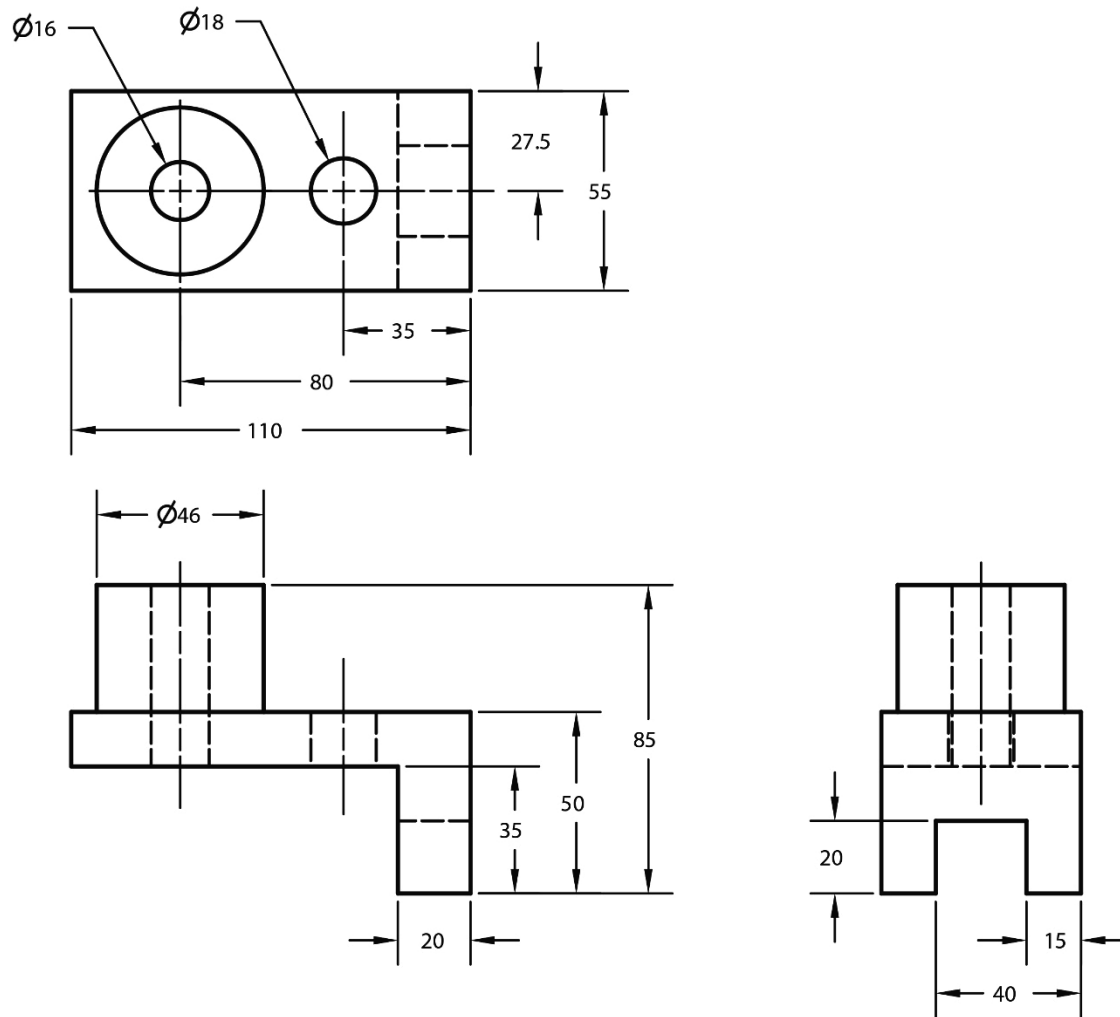UNLESS OTHERWISE SPECIFIED

P2-3) Create a solid model of the following 1020 Steel part and calculate the weight of the part. Dimensions are given in inches.
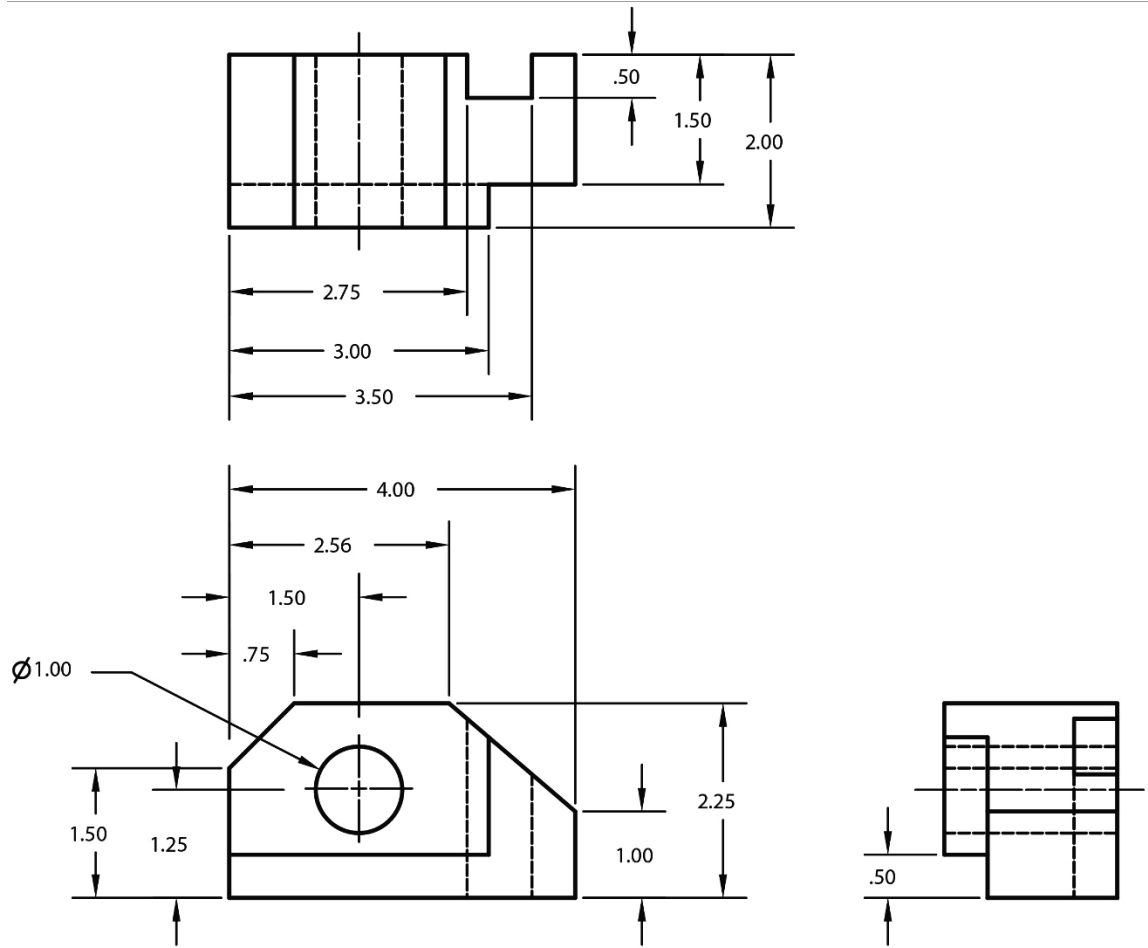
P2-4) Create a solid model of the following ABS plastic part and calculate the weight of the part. Dimensions are given in inches.

4X Ø.50

.62

2.38

.62

4.88

5.50

3.94

3.31

2.19

1.56

.62

3.00

.62

4X .50 X .50

2X Ø1.00

5.13

3.62

P2-5) Create a solid model of the following Brass part and calculate the mass of the part. Dimensions are given in millimeters.
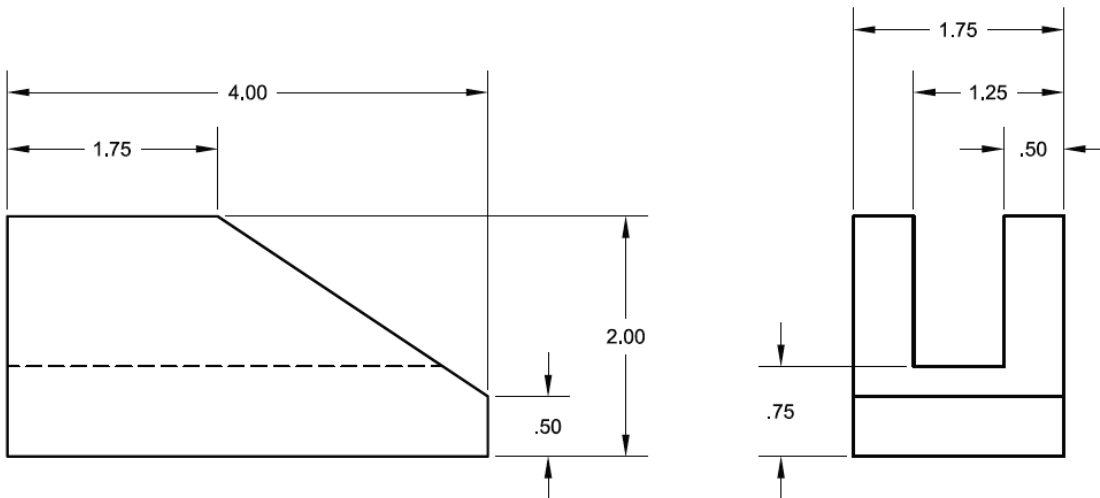
P2-6) Create a solid model of the following Oak part and calculate the weight of the part. Dimensions are given in inches.

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.
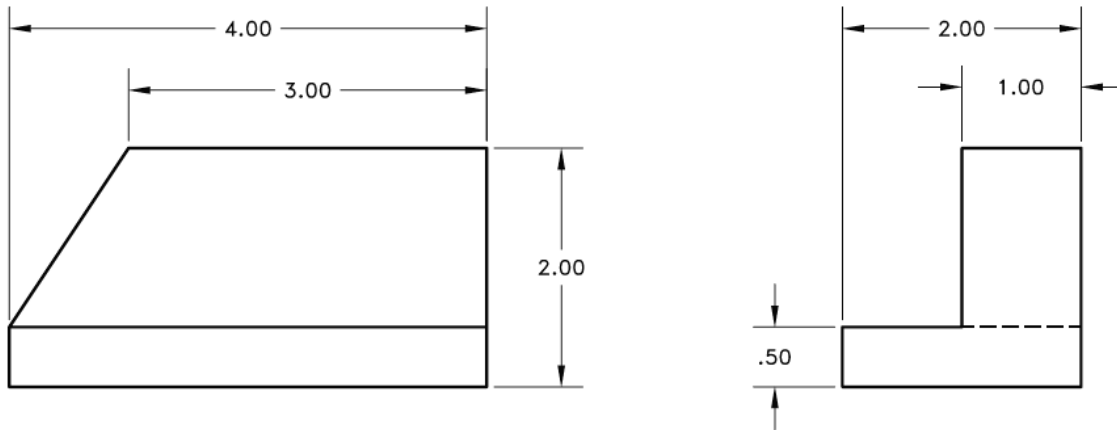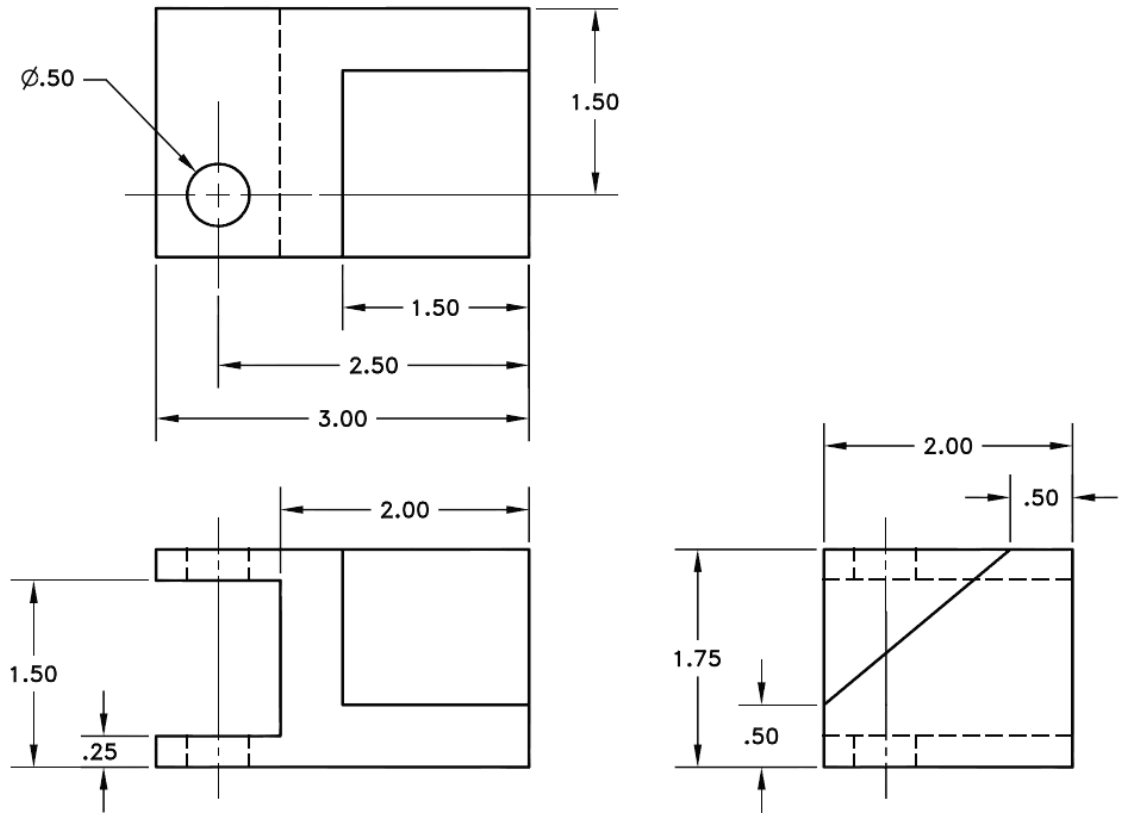

P2-8) Use SOLIDWORKS® to create a solid model of the following 6061 Aluminum part. Calculate the weight of your part. Dimensions given in inches.
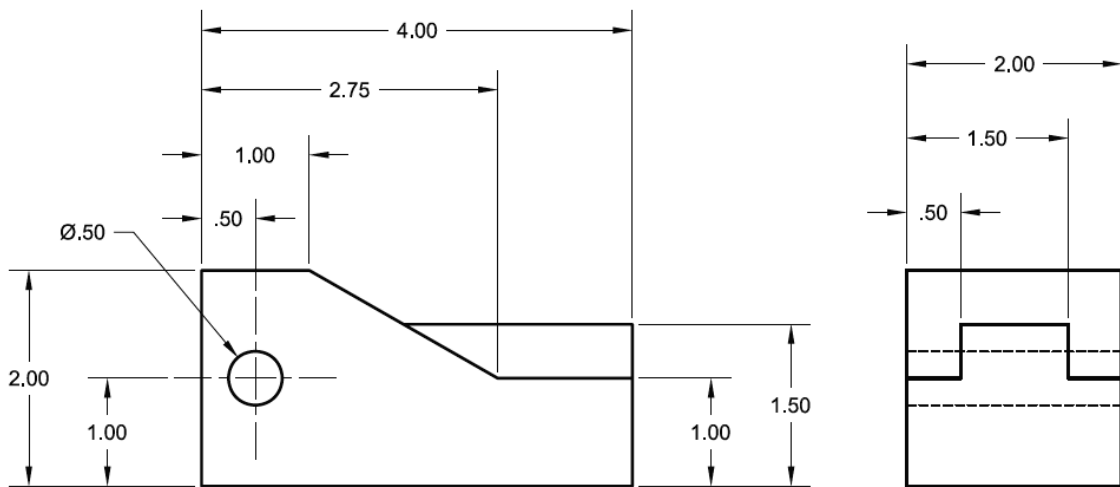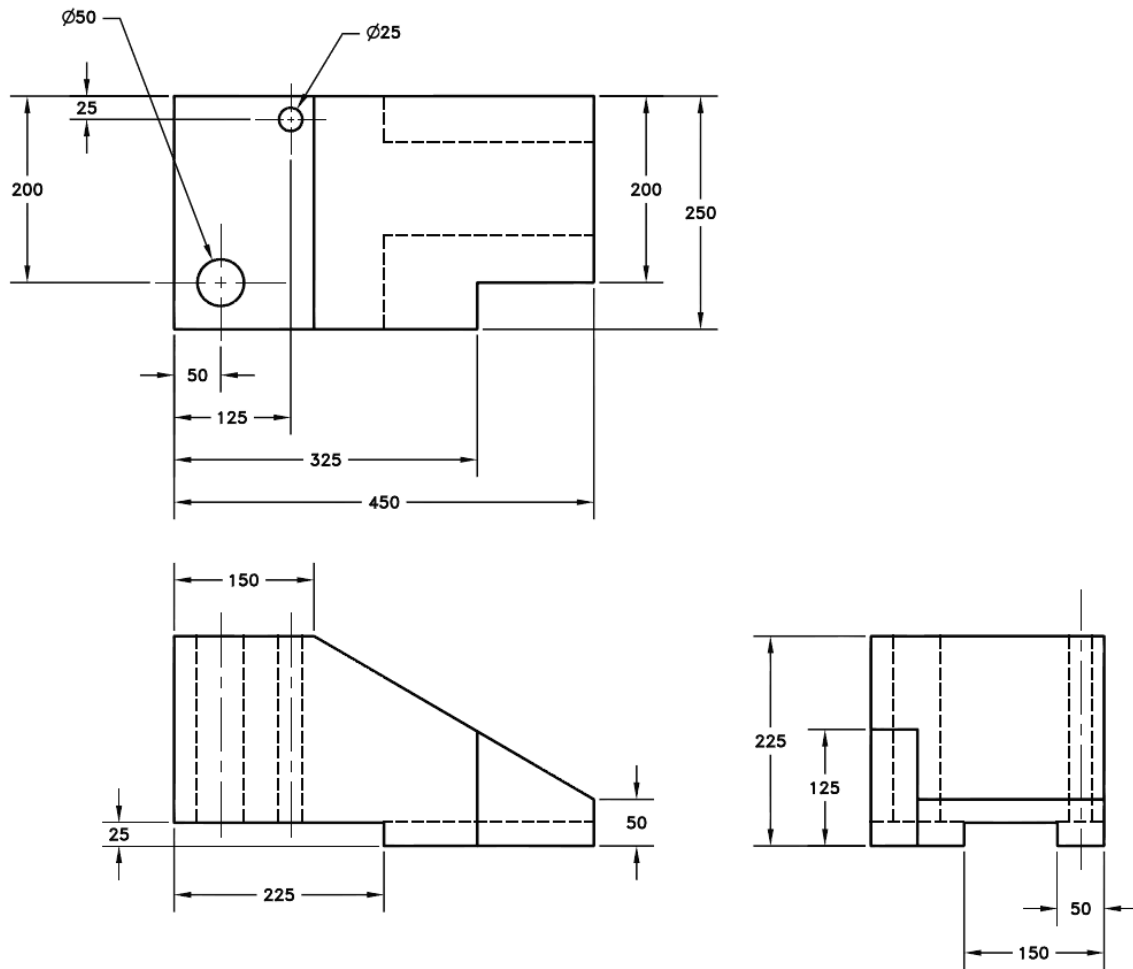
P2-9) Use SOLIDWORKS® to create a solid model of the following 1020 Steel part. Calculate the weight of your part. Dimensions given in inches.
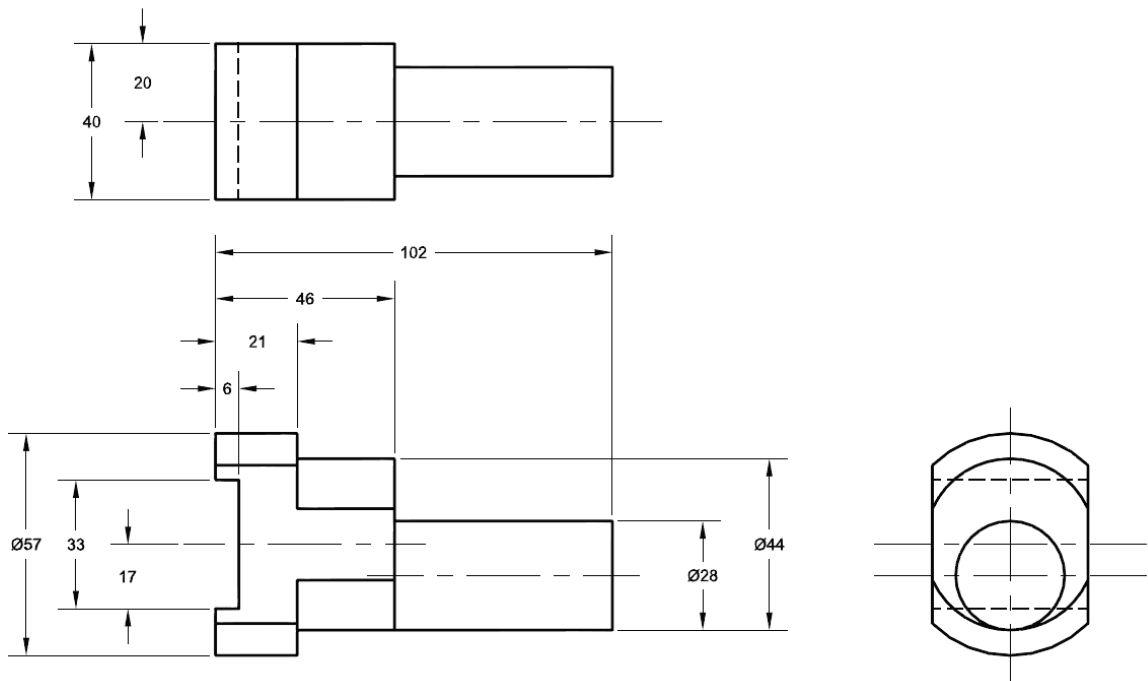

P2-10) Use SOLIDWORKS® to create a solid model of the following 1020 Steel part. Calculate the weight of your part. Dimensions given in inches.
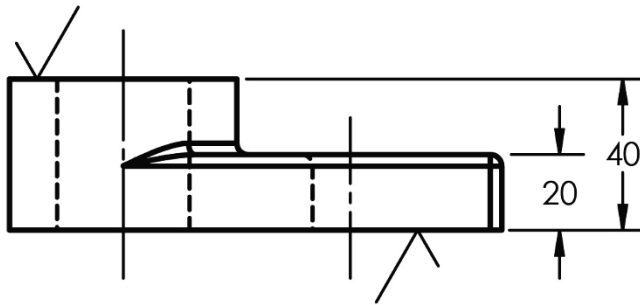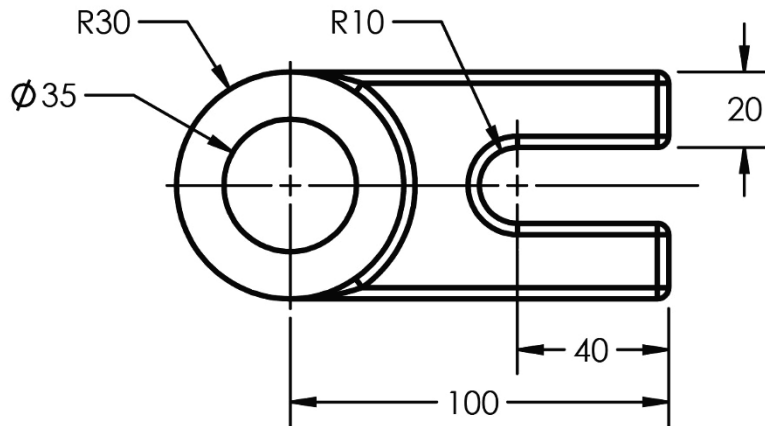
P2-11) Use SOLIDWORKS® to create a solid model of the following ABS plastic part. Calculate the weight of your part. Dimensions given in millimeters.

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.
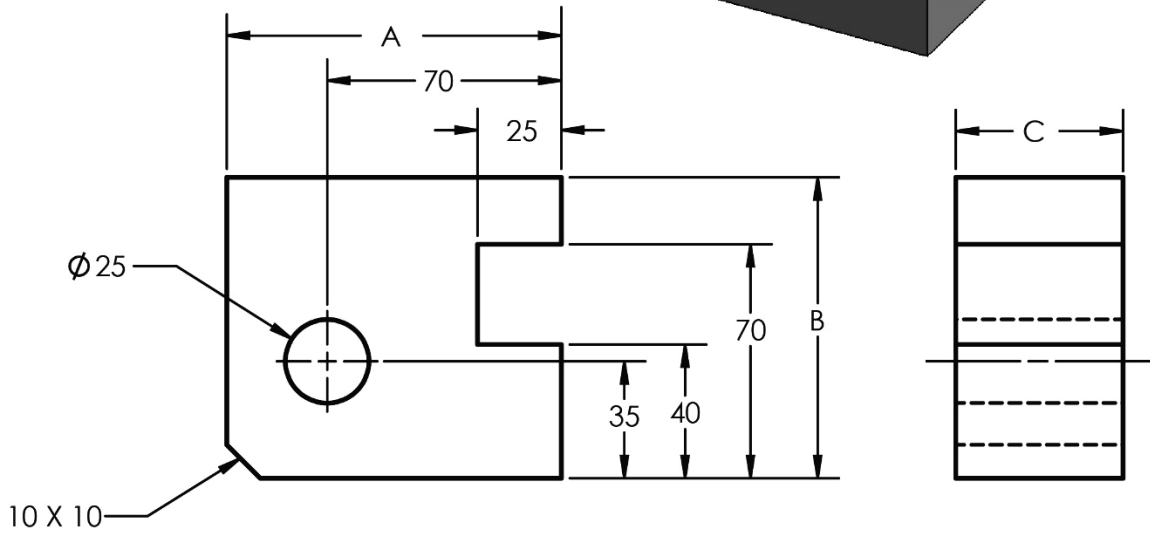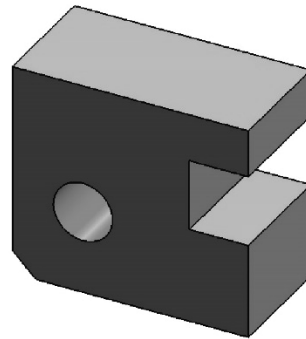
CONNECTING ROD PROJECT (MODEL) QUIZ PROBLEMS

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	Group 2	Group 3
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